C3 CAD-CAM-CNC Computer
Documentation Package

SB Precision LLC
Rev 1.1 May 2009

Thank you for purchasing the C3 computer system. The C3 offers a low-cost integrated cad-cam-cnc controller suitable for nearly any type of CNC mill, lathe, router, or other motion control application.

System Overview

The C3 is a stand alone pc-based design and cnc controller that can be used out of the box or expanded as needed. A standard pc motherboard boots automatically to the Turbocnc machine control program. Step, direction, and Input-Output (I/O) signals are output through the parallel (lpt) port. In order to use the C3 you will need to connect the lpt output pins to your properly-wired stepper or servo motor drives and adjust the configuration settings as required.

Standard g-code programs may either be loaded from other computers into Turbocnc and run, or created using the included Bobcad-cam Gold V14 software. For your convenience printed manuals for Turbocnc, Bobcad, and the Freedos operating system are included. Simple instructions are provided here; please refer to the detailed information in the manuals.

Any standard pc computer component may be added to the C3 as required to support the customization of your machinery. For simple lathes, mills, plasma cutters, and routers no upgrades or changes should be needed.

Basic Operation and Configuration

Attach the included power cable, a USB or PS/2 mouse and keyboard, and your choice of monitor to the connectors at the back of the C3. Press the power button and your C3 will boot to Turbocnc
Connect your stepper or servo drives to the lpt connector. The default Turbocnc.ini configuration file is set for a 3-axis machine with lpt pins as follows:

Pin	Function
2	X-Step
3	X-Direction
4	Y-Step
5	Y-Direction
6	Z-Step
7	Z-Direction

You may make changes to the configuration file by pressing the ALT-C keys. Here you can change axis settings, I/O settings, and other software properties. When changes are made choose the "Reset Ports" option and you will be prompted to save the configuration file.

BobCAD-CAM

To access BobCAD-CAM you need to first exit Turbocnc. Choose File and Exit and you will see the command prompt. Type "bobcad" and press enter twice. You will see the BobCAD-CAM screen appear. Follow the instructions in the Bobcad manuals for help using the system. To return to Turbocnc, type "turbocnc" at the command prompt.

Transferring Files to and from other Computers

A USB flash drive may be used to transfer files with other computers. Refer to the index of Dos commands for more information on how to transfer files using the command line. Most files to be transferred will be from a separate design computer to the C3 computer for machine operation. In this case all the user needs to do is to save the files on the USB drive on the design computer, plug the USB drive into the C3 and then open the file in Editor using the Turbocnc menu.

For advanced users who wish to connect into an RS232 serial network in their shop a copy of Texas z-Modem is included.

Creating and Cutting a Part Using BOBCAD-GOLD 14.1 and TurboCNC

SB Precision, LLC

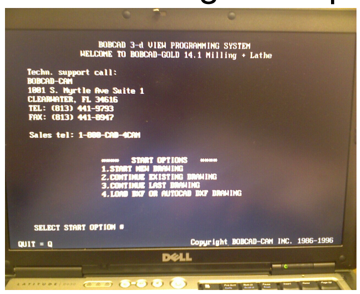
Version 1

sbprecision@yahoo.com

Create the Part Geometry

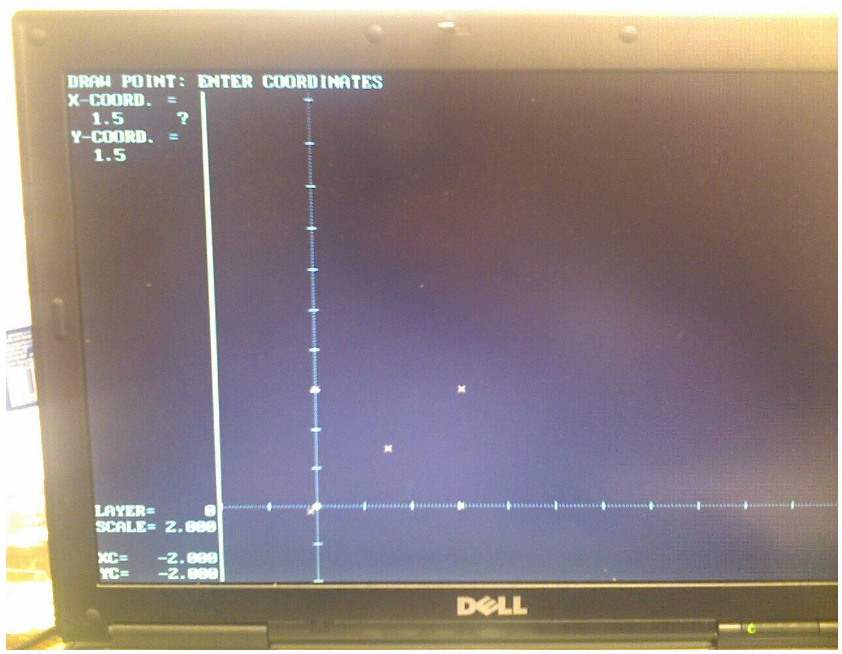
In this tutorial we will create a simple part out of 1/16" sheet that is 3x3" inches square with a 1" diameter hole exactly in the center

+ Make sure the cnc machine is turned off, start the computer and exit Turbo CNC. at the C:\ prompt type "Bobcad" and press enter. At the main menu choose Bobcad-cam. Under START OPTIONS choose "1. – Start New Drawing". Enter a name for the drawing when prompted (such as "test")



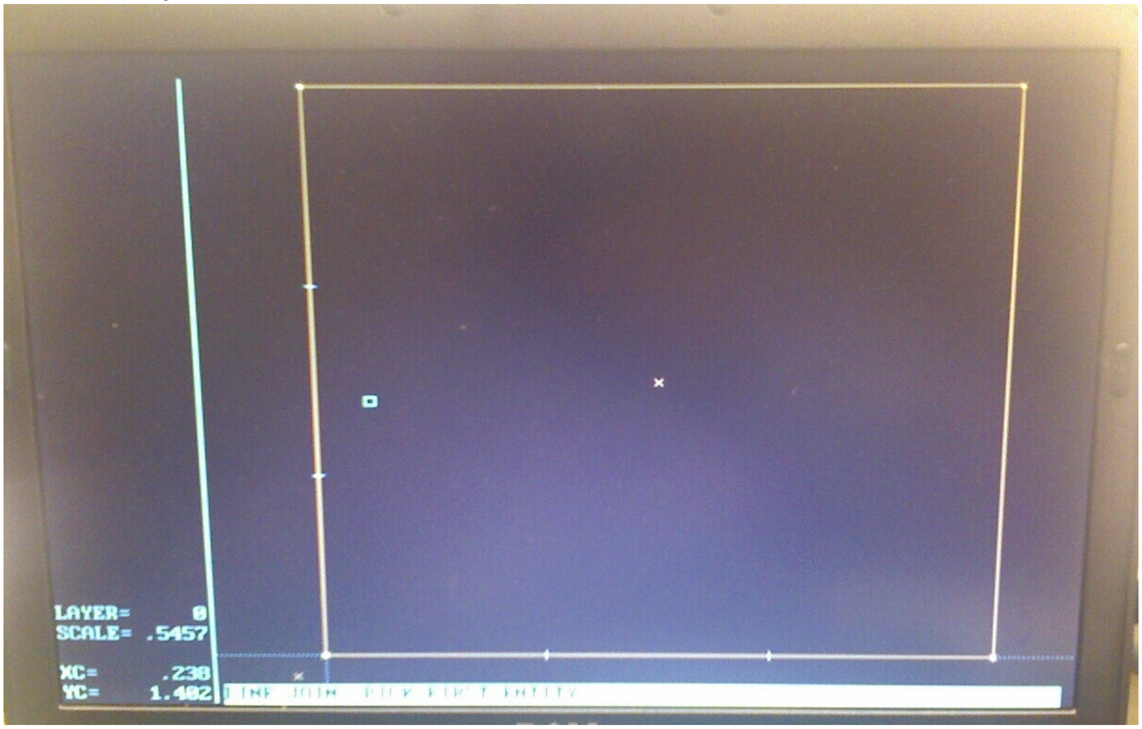
+ Select POINT from the main menu and choose option 2- COORDINATE. Enter the x and y coordinates and draw 6 points as follows:

Point	X	Y
1	-0.125	0.125
2	0	0
3	0	3
4	3	0
5	3	3
6	1.5	1.5

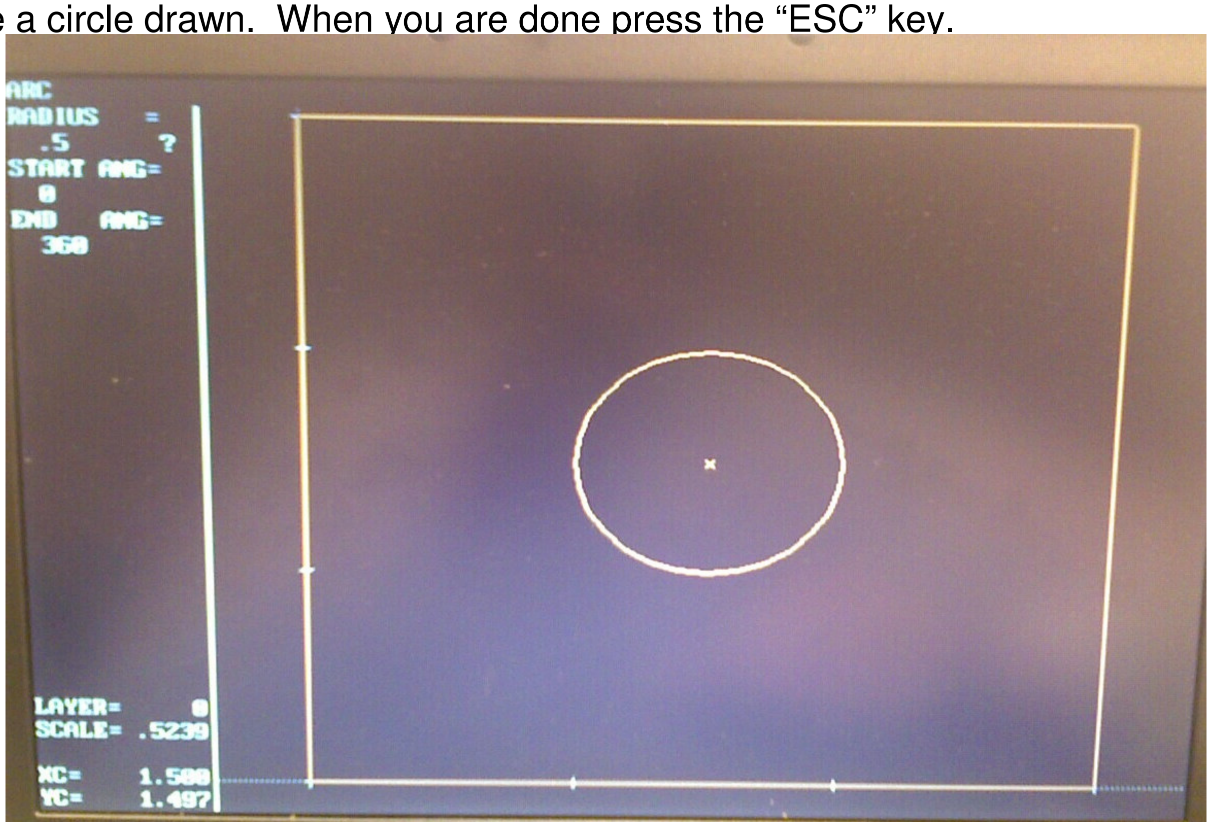


Create the Part Geometry

+ From the main menu select LINE and choose option 3- JOIN. Use the mouse to select the lower right point and then select the upper right point. You will see a line drawn automatically. Repeat for the other 3 sides of the square. Then press the "ESC" key



+ From the main menu select ARC and choose option 2- POINT CNTR. Enter .5 for the radius and then select the point in the center of the square. You will see a circle drawn. When you are done press the "ESC" key.

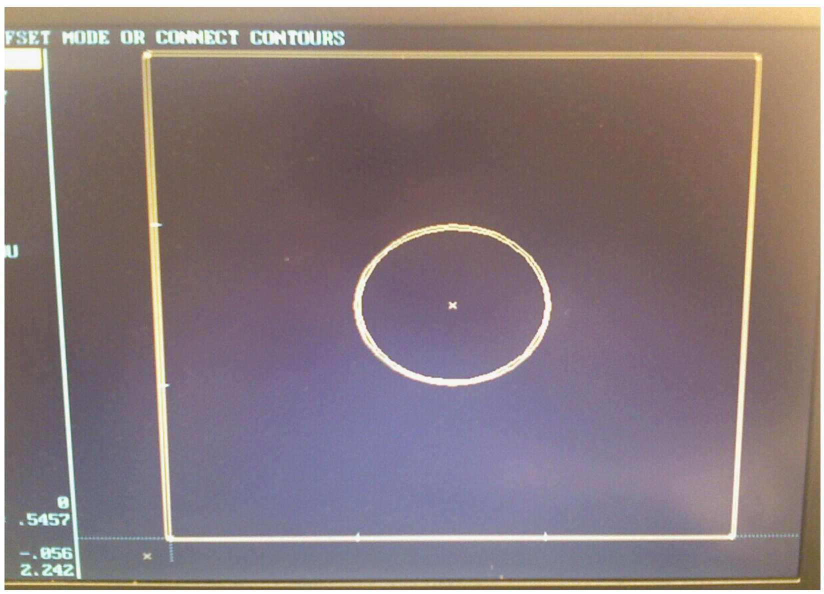


Create the Offset Tool Path Geometry

The cutting tool in this example is .036" wide. We need to trace the geometry of the part by ½ of this width so that the final dimension of the part cut are as close to size as possible.

+ From the main menu select OTHER CURVES and choose option 2- OFFSET. Select SINGLE and enter .018 in the distance. Select the circle in the center of the part and then move the cursor to the inside of the circle and click the mouse.

+ From the main menu select OTHER CURVES and choose option 2- OFFSET. Select CHAIN and enter .018 in the distance. Select the left side of the square that frames the part. Move the cursor to the outside of the square and click the mouse button. Press the "]" key and choose YES. You will now see the offset trace around the part.



+ From the main menu select EDIT and choose DELETE and then SINGLE. Select and delete the original corner points, circle and square.

+ From the main menu choose SAVE to save your drawing.

Create the Tool Paths Using the CAM Function

In this step we will use the NC....CAM functions to create the tool paths file for our CNC controller

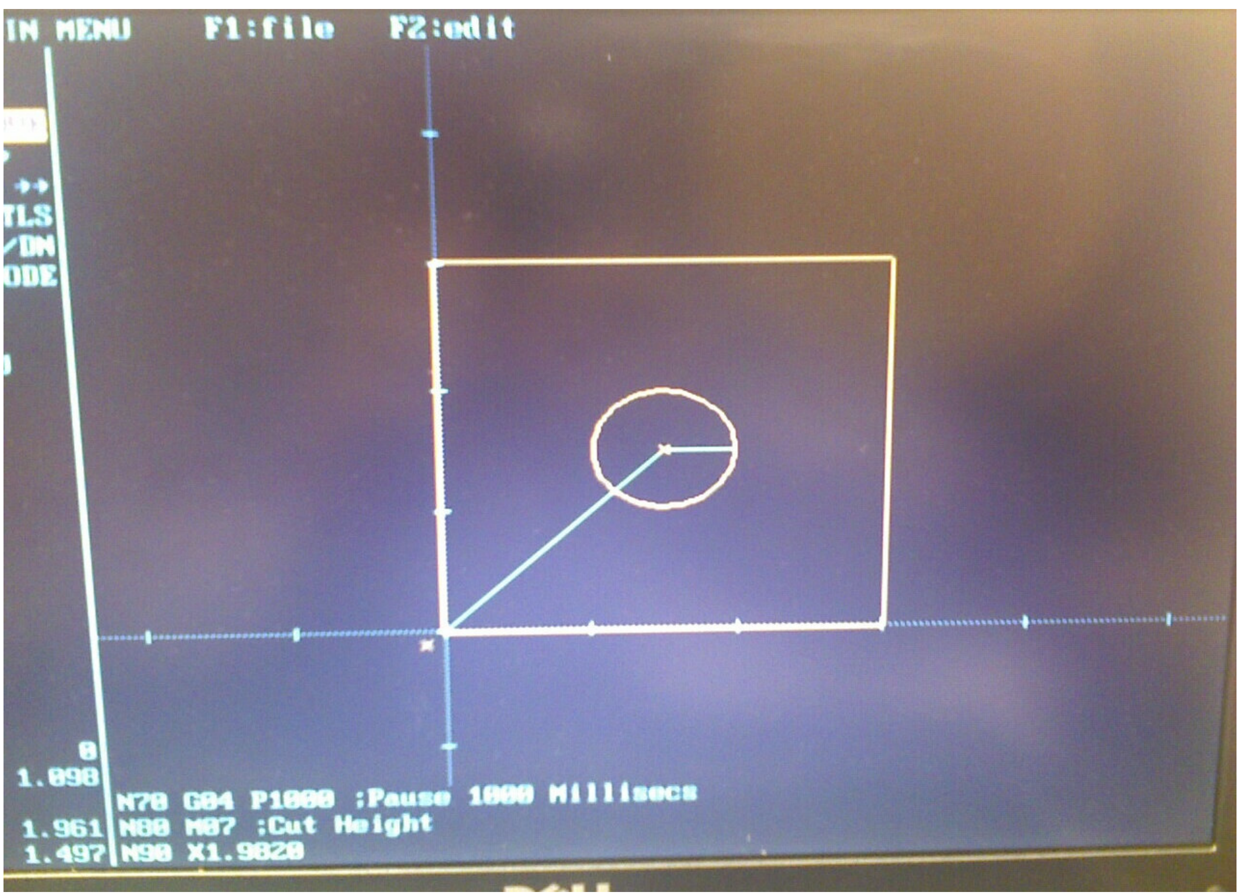
+ From the main menu select NC......CAM and choose NEW FILE. Name the file and then choose TURBOCNC.CFG as the configuration.

+ Select option * RAPID and press enter to change to FEED mode.

+ Select POINT MOVE and move to the center point you created earlier.

+ Choose 7, Tool Up/Down. Move the tool down using FEED mode

+ Choose POINT MOVE and move to the edge of the circle



Create the Tool Paths Using the CAM Function

+ Choose SINGLE, select the circle, and then move the cursor with the mouse until the arrow shows a counter-clockwise direction. Click the mouse button and you will see the circle traced in blue.

+ Choose 7 Tool Up/Down and move the Z axis up

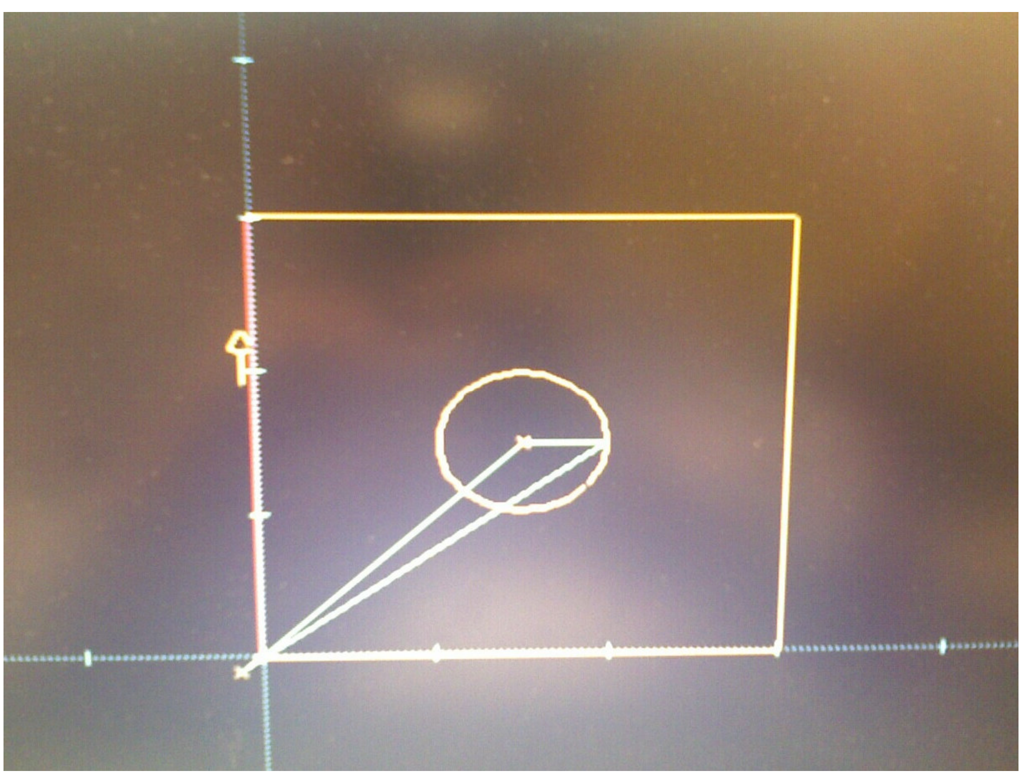
+ Choose POINT MOVE and move to the point your created earlier at -.125X and Y.

+ Choose 7 Tool Up/Down and move the Z axis down

+ Choose POINT MOVE and move to the nearest part of the square.

+ Choose AUTO, select the square, and then move the cursor with the mouse until the arrow shows a clockwise direction. Click the mouse button, press the "]" key and you will see the square traced in blue.

+ From the main menu choose to save the file and exit Bobcad



Examining the Finished Code

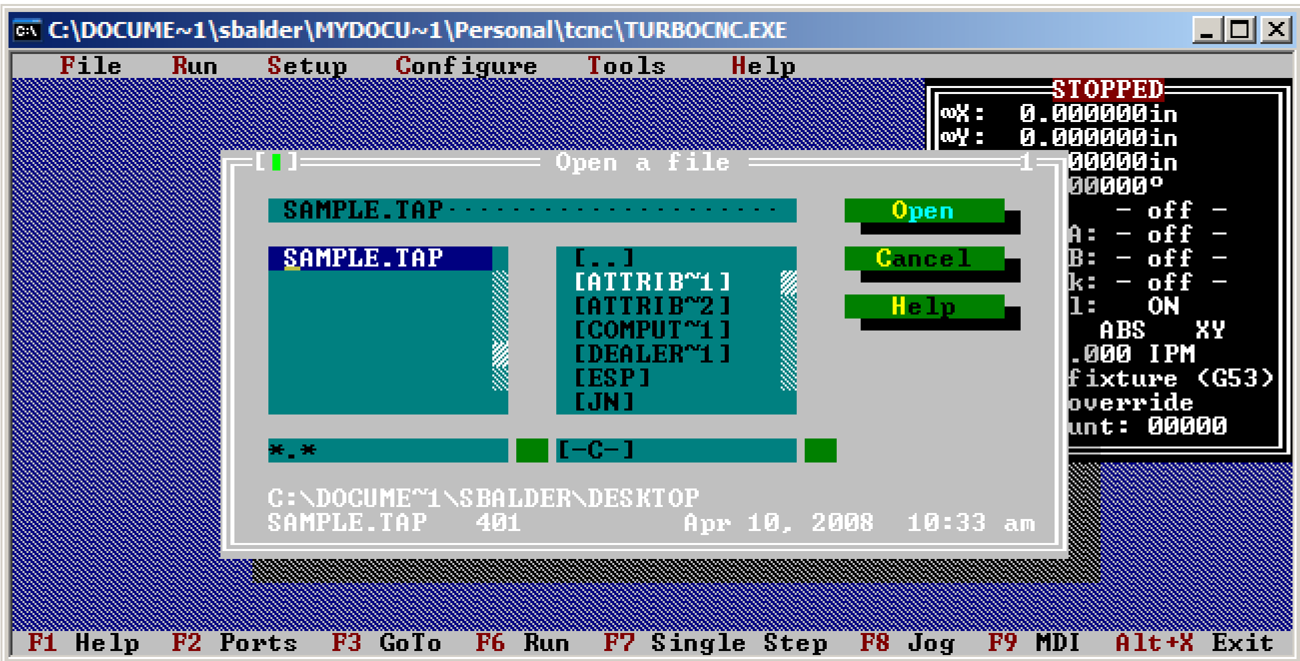
Each line in the finished file is called a Block and has a specific purpose

```
N10 G90 ;Absolute Coordinates
N20 G01 X0 Y0 Z.1 F100 ;Move to 0,0 at F100
N30 G00 X1.50000 Y1.50000 ;Move to Center
N40 G01 Z-0.10000  ;Plunge tool
N50 X1.98200 Y1.49998  ;Move to circle edge
N60 G02 X1.98200 Y1.50000 I-0.48200 J0.00002  ;Cut the center circle
N70 G00 Z0.10000 ;Raise tool
N80 G00 X-0.12500 Y0.12500 ;Move to outside of part
N90 G01 Z-0.10000 ;Plunge tool
N100 X-0.01800 Y0.00000 ;Move to edge of part
N110 Y3.00000 ;Cut in a straight line
N120 G02 X0.00000 Y3.01800 I0.01800 J0.00000  ;Round the corner
N130 G01 X3.00000 ;Cut in a straight line
N140 G02 X3.01800 Y3.00000 I0.00000 J-0.01800 ;Round the corner
N150 G01 Y0.00000 ;Cut in a straight line
N160 G02 X3.00000 Y-0.01800 I-0.01800 J0.00000 ;Round the corner
N170 G01 X0.00000 Cut in a straight line
N210 G00 Z0.10000 ;Raise tool
N220 G01 X0 Y0 Z.1 F100 ;Move to 0,0
N230 M30 ;End program
```

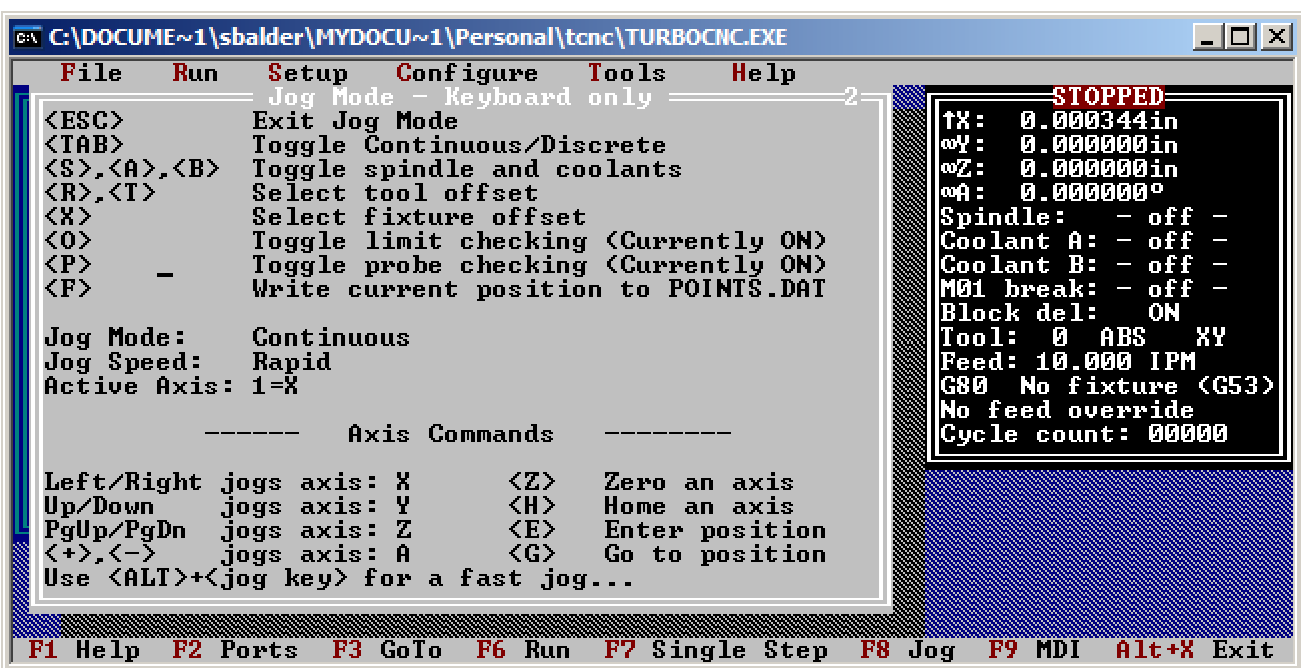
Transfer the G-Code File to the Machine and Cut

Launch Turbocnc by turning the computer on or typing “turbocnc” at the C:\ prompt

Load your program by selecting File ->Open in Editor

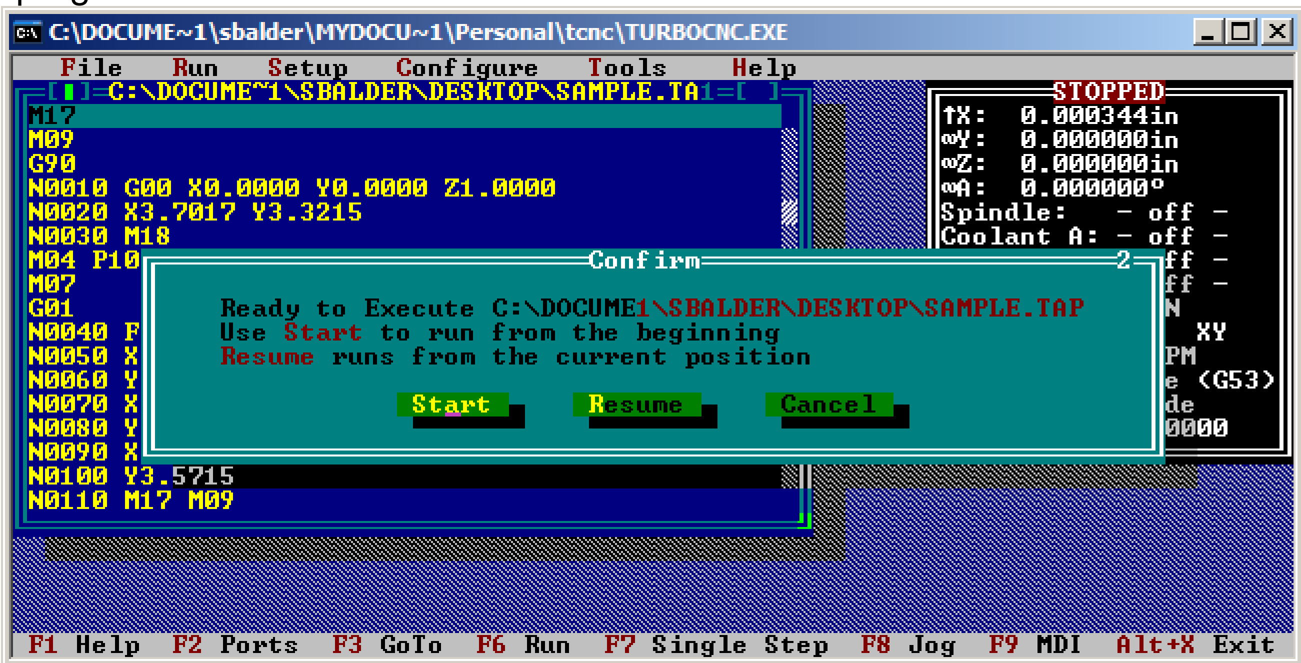


Press f-8 to enter the Jog menu. Use the arrow keys to move the torch to the origin of your workpiece. Next zero the axes by pressing “Z” and then “*”. Hit Esc to exit the jog menu.



Transfer the G-Code File to the Machine and Cut

It is good practice to do a dry run cut of any cnc part prior to actual machining. Raise the Z head well above the part and zero it. Press the f6 key to run the program and choose start.



You will see the program run through each line, and the mill table should move to "trace" the shape of the cut. Correct any code issues before proceeding.

Once the test cut has been made you can cut the actual part. Load the material into the machine, set the Z height to hit the back of the material part to be cut.

When you are confident that everything is in order turn on the spindle. Run the program to make your part.

TURBOCNC V4.01

CNC MACHINERY
CONTROL PROGRAM

Table of Contents

Part 1 – Quick Start

Legal Notice

WARNING !	Various warnings appear throughout this manual. Do not take these to be applicable in all situations, nor to completely describe the hazards involved. CNC machines supply the power to do your work more effectively. **You supply the judgment**. Although DAK Engineering does its best to stand behind this product, we won't be liable for damages incurred.

Regular users of TurboCNC are expected to register the program by paying for it (see contact section). Source code and bug report/upgrade newsletters are made available to registered users via email.

If you're a registered user, you can do anything you want with the program and source code to modify it as you see fit, except for redistribution. In your own shop, anything goes.

Upgrade Information

There have been numerous changes to TurboCNC since the major code branh (version 3.X). These include modifications to the code accepted to bring it into line with the RS-274 D standard and industry practices. G16 is no longer supported. Support for G72 and G73 has been dropped, use G02 and G03 with a third axis callout instead to perform helical moves. Similarly, use M98 as a subroutine call, and M99 as a subroutine return instead of the M60 and M62 supported in previous versions.

New functions in TurboCNC include:

- G76 Multi-pass threading
- G93 Inverse time feed mode
- G178 – Speed Peck Motion
- G183 – Speed Peck Drill Cycle
- M97 – Jump

Some of the functions have had their parameters changed. As an example the '#' is no longer used as a parameter, therefore G04 (Dwell) now accepts the 'Q' parameter to specify dwell time.

Programming elements including a means of communicating with the operator, variables, expressions, and conditional execution, are new to this version of TurboCNC. These additions bring new power and flexibility to your programs.

The format of the turbocnc.ini file is backward compatible, so you can use your old one to get rolling with this new version right away. Inside TurboCNC, use the “Save configuration” option to re-write the turbocnc.ini file, and you’re now up to date!

Since 4.00, there have been scores of bug fixes, and some dialog changes to make the software easier to use. Many of the changes have been “under the hood”.

Support for Peter Homann's DigiSpeed spindle control has been added. For info on this product go to http://www.homanndesigns.com/ .

Installation

System Requirements

486DX2-66 or later PC compatible computer with at least 4MB RAM and a DOS compatible file system

Open 25-pin parallel port for control

500k free disk space (7M for source code and development tools). TurboCNC can be run from a floppy disk, but a hard drive installation is generally better

A 66 MHz or faster clock speed is recommended for satisfactory performance.

Almost any home or office computer system made after 1993 will meet these requirements. However, some fairly modern industrial control computers may not. Consult your owner's manuals to be sure. TurboCNC will report if any critical things are missing at startup.

For very old computers or those that don't have a math coprocessor, try using version 3.0f of TurboCNC instead. This is available in the web archive for download at http://www.dakeng.com/archive.html and although it lacks many features compared to the later versions, it can and has been used for production grade work on even very old 286-10 machines. In some countries these may be the only computers available to private citizens.

A note on Laptops:

Some laptops pose problems to TurboCNC and CNC machines. There are two common problems.

The first is the BIOS may introduce its own interrupts, which can interfere with the generation of a steady pulse train. This problem can cause lost steps. You can try to eliminate these by booting the system into BIOS and resetting the options there.

The second common problem is that some laptops do not switch between +5 volts and 0 volts on the pins of the printer port as is required by many drivers. A parallel port breakout board of your own design or a commercial offering such as the Axxus Technologies DB1V2.0 can be used to restore the full 5-volt swing required by many stepper and servo motor drives.

Program Installation

Here's how to get TurboCNC on your computer. Future versions will feature an installer utility, but for now you have to do this manually.

1. Download a copy of the program from the web at http://www.dakeng.com/turbo.html, and save the file somewhere on your machine.

2. Obtain a de-archiving utility that will handle .ZIP files. Our recommendation is WinZip, which has a 30-day demo and integrates well with Windows – downloadable at http://www.winzip.com. DOS users can use their beloved PKZIP from PKWare or a similar product.

3. Extract the contents of the archive to a convenient folder with a short name, like C:\TCNC\ or similar. You must respect a limitation of eight characters here thanks to an inherited DOS limitation.

4. All of the program files, and this manual, will be found in the new folder.

5. If you want to install TurboCNC on a different machine, copy the contents of the folder to a diskette or transfer it over a network to the new machine. No registry settings, hidden data, or system file changes are used.

Getting Started

TurboCNC runs in real-mode DOS for maximum speed and control over the timing of the parallel ports.

Booting into real mode may be a challenge depending on what system you use currently. Here's how you can get the program started and working efficiently for you under some of the more popular PC operating systems.

WARNING !	If you launch TurboCNC from Windows, you'll have problems when you try to control your machine. Read this section and follow the instructions for your operating system. If you just want to "play" with the program without a machine connected, you can ignore this stuff for now. When it comes time to make chips you'll need this information.

NOTE: These directions assume the program has already been installed per the above.

MS-DOS (any version from 3.0 & up)

Modify your CONFIG.SYS file to allow a boot-up configuration in which EMM386.EXE, HIMEM.SYS, RAMDRIVE.SYS, and any disk caching programs are not present.

Alternately, create a boot disk with the FORMAT /S command to boot up in a minimal environment.

Disable DOUBLESPACE if it is enabled.

Change to the directory with turbocnc.exe and execute the program.

Windows 3.1

Edit the WIN line and any drivers from your autoexec.bat and config.sys files in order to present a clean boot to DOS.

Run TurboCNC from the DOS prompt. If you're still using Win3.1 this far into the 21st century, we assume that you know what you're doing here. Generally, the instructions are the same as for MS-DOS above.

Windows 95/98

From the desktop, pick Start | Shut-down | Restart in MS-DOS mode

Change to the directory with turbocnc.exe and execute the program using the DOS commands

CD \TCNC or similar for your system, then TURBOCNC

Alternately, you can hold down the CTRL key (or sometimes F8) while your computer boots up. A short text menu will appear with some boot options. Pick Command Prompt Only, and run as above. If TurboCNC gives you an EMS/XMS driver warning, do it over and pick Command Prompt Safe Mode instead.

Windows NT / 2000

TurboCNC will not drive your CNC system reliably as direct access to the hardware is not allowed under these operating systems. You can, however run TurboCNC under the command prompt to familiarize yourself with it, and to 'Dry Verify' parts files. All of the screen shots in this document were gathered by running Turbo CNC on a Windows 2000 system and performing a 'Print Window'.

TurboCNC can run on a dual boot system with DOS. Consider formatting a disk partition with FAT16 rather than running solely from a floppy disk. On some systems using floppy drives only TurboCNC has had problems due to drive access times.

Windows XP

You'll need to create a real-mode boot disk for TurboCNC Get a blank floppy disk out.

- From the desktop hit Start | My Computer | Floppy (A) [or other letter as appropriate for your system]
- Insert the disk, then hit File | Format
- Check the box that says "Create MS-DOS boot disk" and hit OK to format

Expand and copy the TurboCNC files to the floppy.

Reboot the computer with the floppy in drive A

Execute TurboCNC at the A:\> prompt.

Contact Information

The best way to contact us is via email:

staff@dakeng.com

Or if you prefer the normal mail:

DAK Engineering c/o Dave Kowalczyk
11032 SE 224 PL
Kent WA 98031 USA

Registration payments ($60) can be sent through PayPal to our account at admin@dakeng.com, or by check/money order to the address above. Make checks payable to **DAK Engineering**. Include your email address so that we can send the source code to you as well.

Support

Consider joining the Yahoo! TurboCNC User Group at http://groups.yahoo.com/group/turbocnc/. Many of our members are highly knowledgeable and willing to help, and several have posted their enhancements to TurboCNC.

Don't hesitate to let us know about features you want to see in a future version. Upgrades are continuous, and most suggestions find their way in there eventually.

For bug reporting, please send the problem code and your turbocnc.ini file for the program as a courtesy if appropriate to the issue. It helps enormously in analyzing problems.

Credits

Dave Kowalczyk – Lead programmer, original author.

Jerry Jankura - Programming, TUI systems and interfaces.

Tony Groothuizen – Programming, debugging.

George and Andrew Bean - Authors of the TechnoJock Toolkit, which drives the menu system.

Terry King - Author of Fkeybit.

Harald Geier - Menu usability, MasterCAM posts.

John Johnson - M60/62 (now M98 / M99) and parsing algorithms.

Daniel Barber - Windows XP compatibility testing and boot instructions.

Alan Matheson - Metric mode testing.

Daniel Brock, Wayne Hill, and Andrew Erwood - G76 cycle specifications.

We would also like to recognize the registered users and the beta testers especially for their support, suggestions, patience, and the many successes they've enjoyed while using this software.

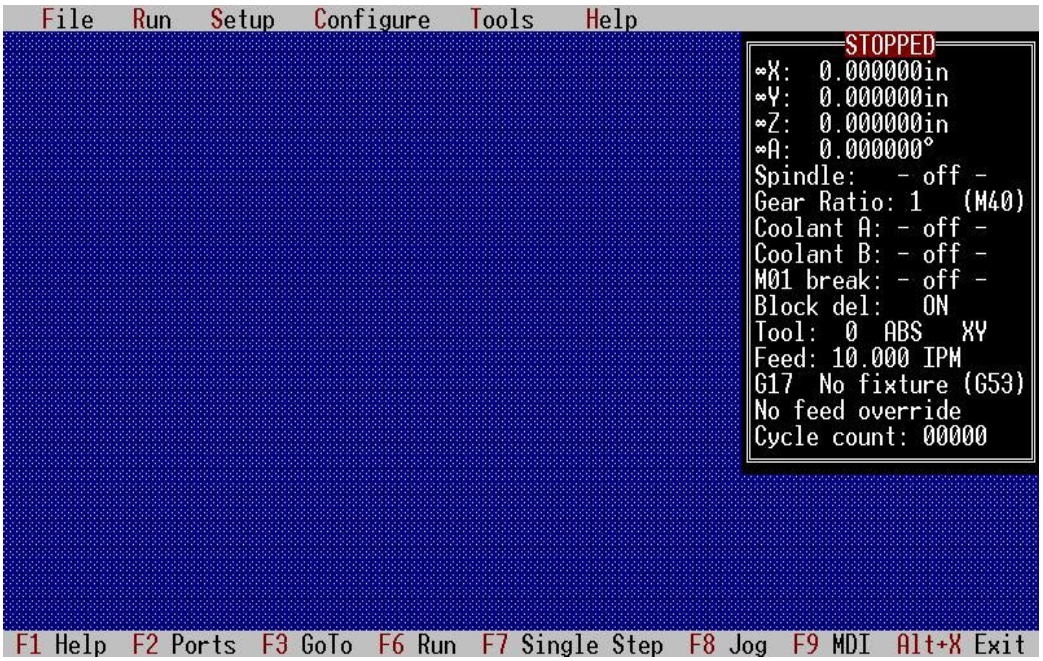
Part 2 – User’s Guide

Introduction

TurboCNC is a machine control interpreter. By loading in “g-code” files and executing them, physical motion of a machine occurs.

Menu System

Here's what you should see when you start the program, after the initial diagnostic screen:



In many respects this program functions like others you have used, such as the concept of opening and editing files, saving, and in the GUI concept.

Note the black colored window on the right hand side of the screen. This is the Status Window, and it's very special. From top to bottom, the Status Window displays the current machine position, the status of the spindle and coolants (if installed), which options are currently active, and some information about the machine state. You can't move this window or get rid of it - it's there permanently.

The position of each axis is updated at the end of a move, and "In motion..." will be displayed while things are still moving. In previous versions you were allowed to have the position updated each step, but this costs too much CPU time to keep up with. The information here is updated just after a block executes.

TurboCNC has been redesigned to allow the use of a mouse. The mouse is deactivated during motion to prevent interference of the mouse driver with the generation of the pulse train. If you experience lost steps while using the mouse, try booting your system without the mouse driver.

TIP: Although the menu system is designed to be used with a mouse, there are keyboard shortcuts for every function. For example, Alt-F-X exits the program. This is usually much faster than using the mouse, especially if you don't have one. In many shops, the amount of dirt and crud around (not to mention the tendency for any horizontal surface to become occupied), will preclude the use of a mouse anyway.

Quick Keys

The functions most commonly used to set up a job have an associated function key listed on the status bar at the bottom of the screen. Other common tasks also have a '**Quick Key**' associated with them. These keys are:

- ctrl + N Open a new file
- ctrl + O Open file in editor
- ctrl + R Run from disk
- alt + num Configuration menu for that axis

Motion Keys:

During motion several keys are checked every 18.2 milliseconds (about 55 times per second). These are:

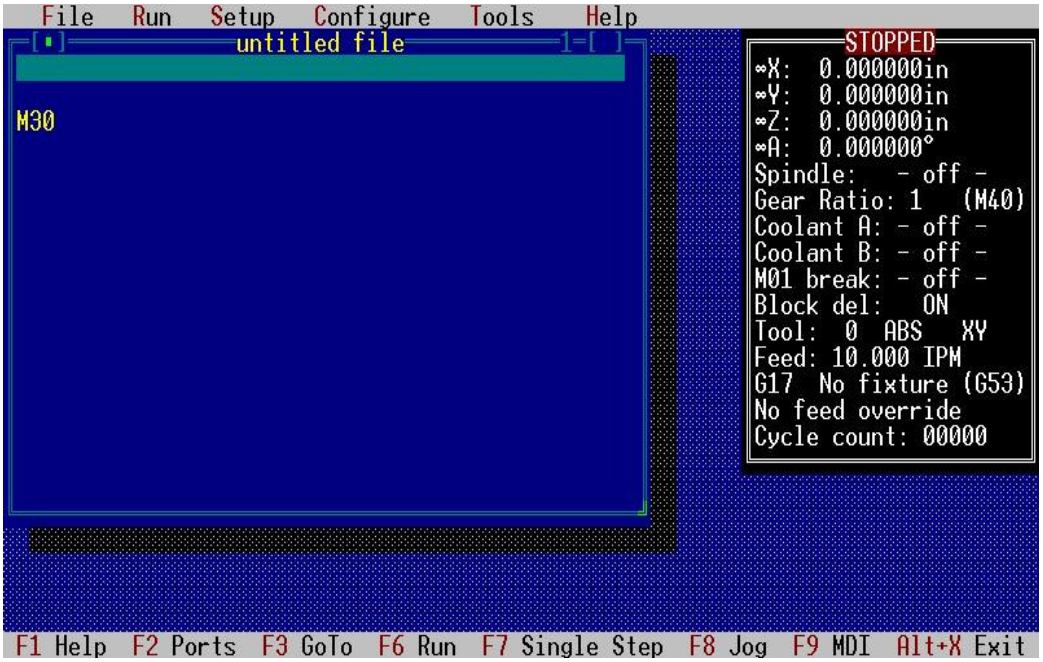
- Esc Panic Stop (Stops motion immediately)
- increase or decrease feed rate 1%
- shift + <>   increase or decrease feed rate 10%

During motion, the result of increasing or decreasing the feed rate will not be seen on the status display. This is updated at the end of the current move.

File

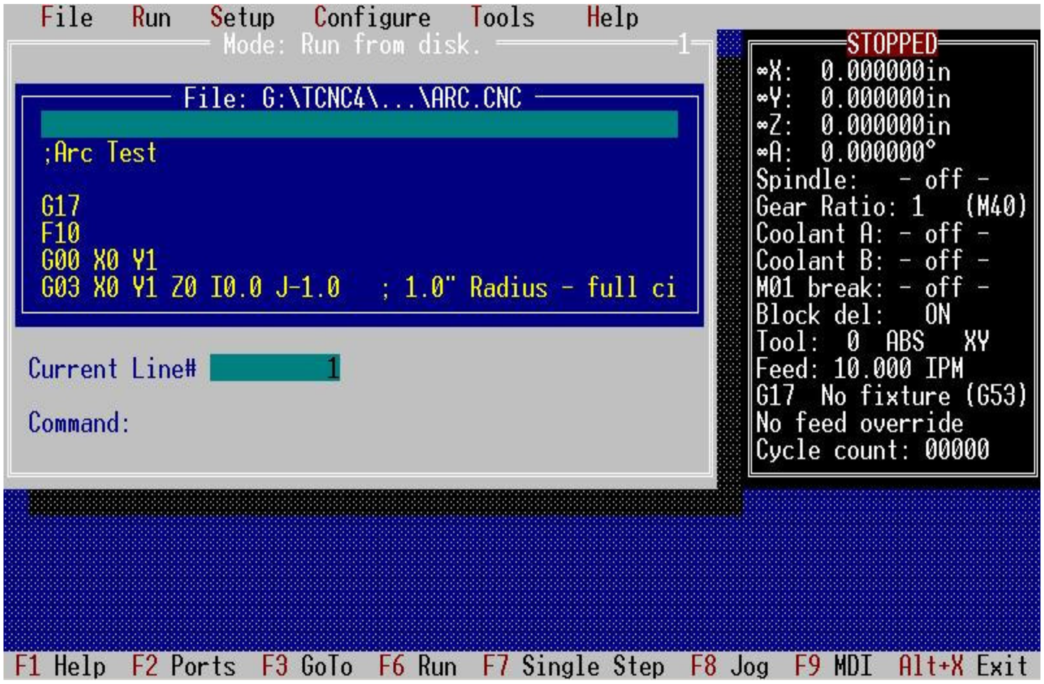
New, Open in editor, Run from file, Close, Save, Save As

These first five options are for accessing and manipulating your collection of g-code files. Note that you can only have one file open at any time. G-code files should be straight ASCII text in CRLF (DOS) format.



Open in editor will load the entire file into memory, and bring up an editing window. This is best for smaller files of 500KB or less. Standard editing keys are available.

- ctrl + c — copy
- ctrl + v — paste
- ctrl + x — cut
- up-arrow — move cursor up one line
- ctrl + up-arrow — move cursor to top of window
- down-arrow — move cursor down one line
- ctrl + down-arrow — move cursor to bottom of window
- left-arrow — move cursor left one character
- ctrl + left-arrow — move cursor left one word
- right-arrow — move cursor right one character
- ctrl + right-arrow — move cursor right one word
- page-up — scroll screen up one page
- page-down — scroll screen down one page
- home — move cursor to beginning of line
- ctrl + home — move cursor to beginning of file
- end — move cursor to end of line
- ctrl + end — move cursor to end of file
- F5 — stretch window
- ctrl + f — find string
- F3 — find again



Run from file is used for large files that won't fit into memory. Editing and scrolling through the file are not implemented in this mode. A console is brought up containing more status information and a display of code being executed. The next line to be executed will turn red if execution tries to continue beyond the end of the file. Code that has been executed will turn gray, pending code is yellow. Most program messaging will occur in the display area of the console, rather than bringing up a separate dialog box in this mode.

Print

Sends the currently open file to the printer.

WARNING !	Ensure that your printer is connected, and your CNC machine is turned off to use this function. It is possible for the standard print function to cause movement and turn on your spindle if the port definitions of your CNC machine match that of a printer.

Load Tooling File

Loads a tool and fixture offset file from disk.

Save Tooling File As…

Saves the current tool and fixture offsets to disk. Expert users will find it generally fastest to edit the tool offset file directly to adjust for tooling changes.

Exit

Exits TurboCNC. You will be prompted for confirmation.

Run

There are a variety of functions to actually do some real work with your machine located under the Run Menu.

The '<' and '>' keys can be used to adjust the feed rate while the CNC machine is in motion. Using the Shift key in combination with these will yields a finer degree of control.

The 'ESC' key functions as a **PANIC STOP**, while in this mode.

WARNING

!

A Panic Stop initiated by the operator immediately ceases the generation of step pulses for the motors, and opens the spindle motor relay (if fitted). The drive enable lines (if fitted) are not set to the disabled state to prevent further injury or damage which may be caused by motion due to gravity. Coolant status is not changed to prevent further injury due to burns caused by hot materials.

After the 'Esc' key has been pressed, or the Emergency-Stop has been activated the operator must select 'OK' in the 'Confirm Motion Abort' dialog box. He or she must then acknowledge that the program has been aborted before taking control of the machine via the MDI mode of operation to change the status of drive enable and coolant states.

NOTE: Panic stop and limit switches are wired to the parallel ports as logical inputs. This is to get around the keyboard buffer in case of an emergency, and allows for fast polling of the input states. After a panic event, the options of continuing where machining was interrupted or aborting completely are available. The former option is good for fixing simple things, like an incompletely tightened tool or something that was noticed just before the "rubber met the road", so to speak. A further option of jogging the machine is available in the 'Run from disk' mode of operation.

Single Step

Single Step is used to step through a new program line-by-line to check for "sticking points". Keep selecting this menu item (or the press the F7 key) to execute a program one line at a time.

Single Cycle

Runs through your entire program once. This is used to make a single copy of a part, usually while proofing out a new program.

Piecework

This option is used to make multiple copies of a part. It pauses between parts to allow new blanks to be mounted, and is the usual method for chucking operations where the operating sequence is one of "load, run, stop, unload, load" by a human operator. A running count of cycles executed is kept in the status window.

Hitting a key starts machining operations from the first line of your parts file again… and again…and again

Automated…

This option is used for fully automated machines that are capable of changing the work piece. Just enter the desired number of parts and send it on its way. It will run the parts file over and over again for the specified number of cycles. This is great if pallet changing, bar pulling, PLCs, or other robotic hardware is available to do the loading and unloading. The G codes to run the robots as other axes must be included in the same program with the code used to machine the part. This is partly the reason so many axes of motion are included.

Dry Verify

This option allows you to run through the file without moving your machine or turning anything on. It is often used to see if there are errors in the file syntax, and to obtain an estimated running time for the program. The estimated run time will be a bit on the low side as Dry Verify uses some approximations for speed.

The ***Reset File*** and ***Go To Block/Cursor*** options allows the operator to establish the current execution point in a file without cutting or moving anything.

Reset File

The ***Reset File*** option is used to reset the program counter to the first line in the file.

Go to Block/Cursor

This is very convenient method of setting the program counter to any line of a parts file. The block may be specified either as the line number of the file or the N-Word on the target line. When resuming machining from a position in the middle of a file, TurboCNC will ask some questions about how to get the machine where it should be if there's a discrepancy in the physical mode or position from what the file expects.

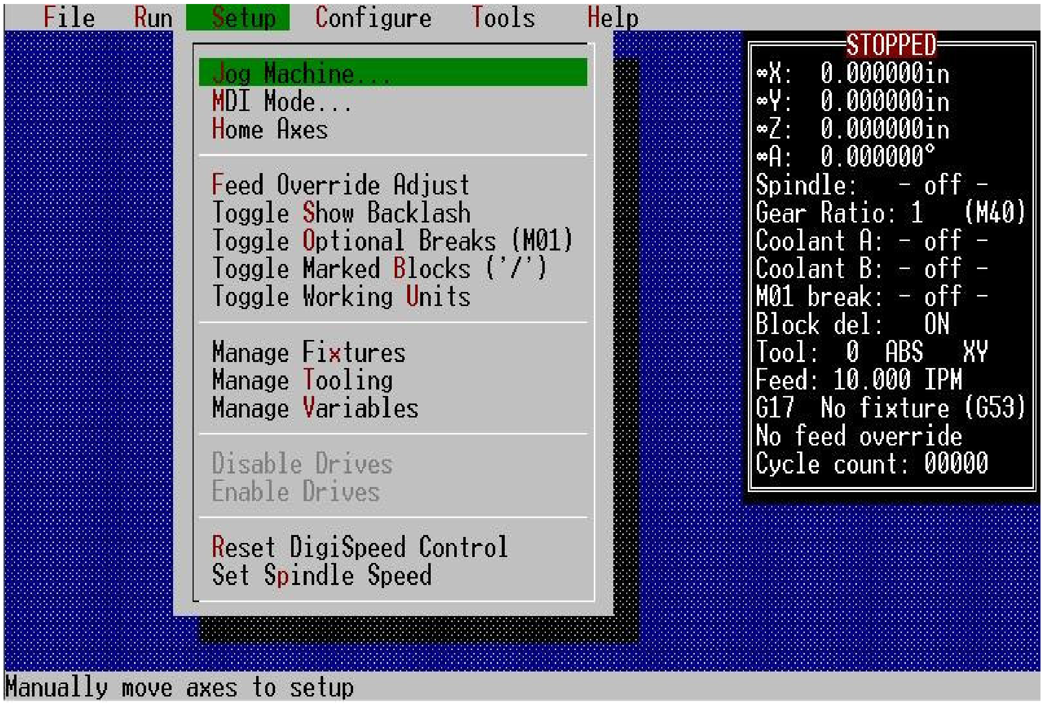
WARNING **!**	Variables are not computed as TurboCNC scans a file to arrive at a specific block. Subroutine calls and jumps may have altered the sequence of execution. **TIP:** Use 'Manage Variables' under the 'Setup' menu or the MDI mode to set variables to desired values before continuing with program execution.

Set Cycle Count

You can set the cycle count to any number from 0 to 99,999 with this menu option. This is typically used for keeping track of production volume, and in some cases for establishing part serialization.

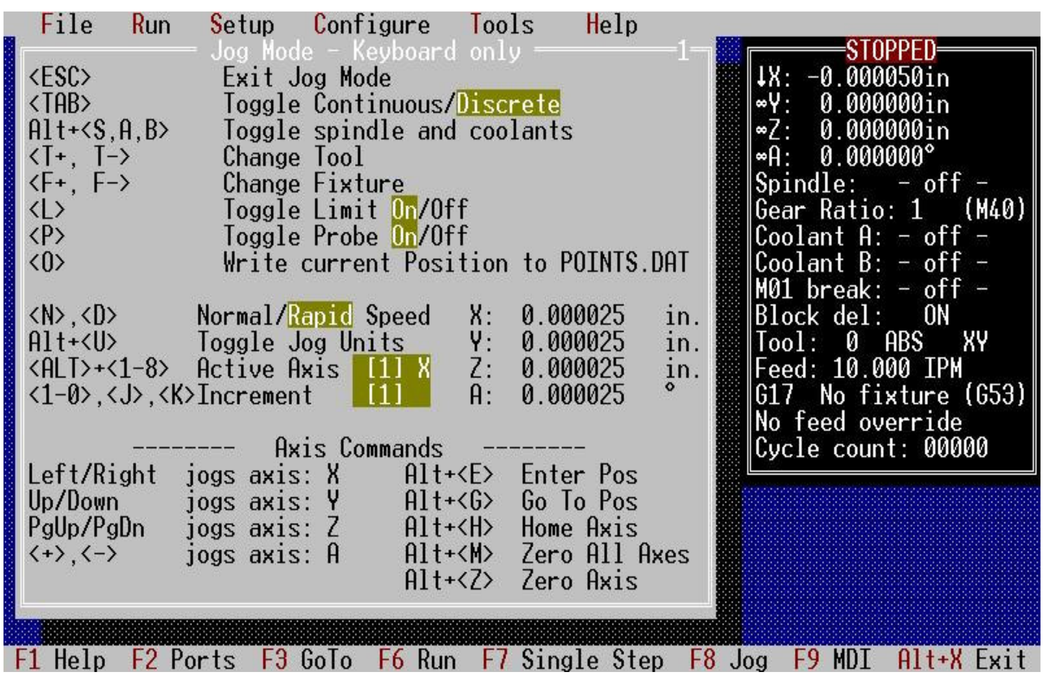
Note that each time a g-code file is run to completion (Single Cycle, Piecework or Automated modes), a “joblog.txt” file is updated in the current directory with the time/date stamp, cycle length, and the file name. Keeps track of your billable machine hours.

Setup



Jog Machine

In Jog Mode, keys are assigned functions that move axes and to turn the spindle and coolants on and off. This mode is used to zero tools to the work in preparation for cutting, or for testing the machine. Generally, actual machining is not performed in this mode. All of the tool and work offsets are available while jogging. The ***Jog Machine*** menu is shown below.



TurboCNC's jog mode offers continuous and discrete jog modes. When in "continuous" mode, pressing a jog motion key causes the respective axis to move at the "slow" jog rate until the key is released. When in the discrete mode, pressing a jog motion key causes the respective axis to move a specified distance and then stop.

The Tab key is used to toggle between continuous and discrete jog modes. The setting is persistent and is automatically saved in the initialization file when TurboCNC exits.

Continuous Mode

In the continuous mode, the jog is frequency based, and continues as long as the jog key is held down. Backlash compensation is applied, if required, and the axis is accelearated, to the step rate specified in it's configuration menu when the key is first pressed. Upon releasing the jog key, the axis is decelerated and stops upon reaching the 'Start Vel' in it's configuration menu. Holding down the ALT key selects the fast jog rate.

Discrete Mode

In the discrete mode, tables are used to specify the distance that the axes should move. The tables are shared between all of the axes. There is a separate increment table for each System of Measurement. A separate index into each table is maintained for every axis. The indicies for all of the axes can be locked together or synchronized. This is indicated by 'Sync' as the active axis, and results in the same incrermeent being used for all axes. The indicies for each axis can also be locked across all systems of measurement. Depending upon the increment specified increment sizes, this will result in jog of equal size in both linear systems of measurement. These options are available on the 'General Configuration' menu.

The J and K keys are used to move the increment index up or down. The index will not wrap from the lowest to highest or highest to lowest values. The numeric keys (1 – 0) can be used to directly specify index 1 through 10 for the active axis without cycling through each increment.

The active axis is specified with the <ALT> + AxisNumber keys, if these are not synchronized. While jogging, the active axis is automatically updated to reflect the last axis moved.

Either the Imperial or Metric tables can be used while working in either system of measurement. This is initialized to the current Working System of Measurement when entering jog. The <U> key toggles the table in use.

A quadrature encoder wheel may be used for input while in the discrete mode. Any move size under .250" will be taken for each pulse from the wheel. Set the Jog Encoder A and B inputs up under the IO config menu to enable this.

When limit checking is on, you will only able to jog as long as no limit switches are triggered. Once a limit is reached, disable limit checks to move again.

When probe checking is on, jogging will stop when a probe input is triggered (electronic edge finder).

Notes:

- The direction of the jog for an axis can be reversed using the 'Jog Keys Invert' option located on it's Configuration Menu.
- The speed at which TurboCNC homes the axes is configurable under the Configure / General menu.

- Keys are configurable for foreign language keyboards. See the ***TurboCNC Configuration File*** section of this manual for details.
- Several of the commands make use of a "***currently active***" axis. You can set the currently active axis by specifying either its description or its number. The "***currently active***" axis is modal. It remains active until another axis is set active. An example – to home the X axis and then set its location to 15 and then move X to 20, you would enter the following sequence:
    - X
    - Alt+H
    - Alt+E (and then enter 15 in the dialog box)
    - Alt+G (and then enter 20 in the dialog box)

  The Y and Z (and any other) axes will remain in their current locations; only X will move and have its location changed.

Key	Function
Alt+A	Toggle Coolant "A" on and off
Alt+B	Toggle Coolant "B" on and off
A	Set Axis "A" as the currently active axis
B	Set Axis "B" as the currently active axis
C	Set Axis "C" as the currently active axis
Alt+E	Set the current location of the currently active axis to the specified value. This command uses G92.
F	Set the fixture number to its next higher value. If the fixture is at its maximum, set the fixture to "no fixtures"
D	Set the fixture number to its next lower value. If no fixture is currently in use, set the fixture number to the highest allowable fixture
Alt+G	Move the currently active axis to a specified position. This function is similar to code G00.
Alt+H	Home the currently active axis
J	Decrement the index into the active speed table for the currently active axis, regardless of synchronization. If the current index value is 1, J has no effect.
K	Increment the index into the active speed table for the currently active axis, regardless of synchronization.. If the current index value is 10, K has no effect.
L	Toggles the Limit switch check
Alt+M	Set all axes current position to be 0.
N	Sets discrete jog velocity to "normal" – the maximum velocity that the jog attains is set by the "F" code
O	Write current coordinates of all axes to point file – used when probing an object.
P	Toggle probing mode. When probing is turned on, sets the velocity for probing.
Q	Set the axis which will be assigned to the + and – keys for jogging
R	Selects Rapid jog rate when the machine is in jogging mode
Alt+S	Toggle the spindle
T	Change tool number to the next tool bin (number). If the tool bin is at its maximum value, wrap to the lowest tool bin
R	Change tool number to the previous tool bin (number). If the tool bin is at its minimum value, wrap to the highest tool bin.
Alt+U	Toggle the active discrete jog table between Imperial and Metric units
U	Set Axis "U" axis as the currently active axis
V	Set Axis "V" axis as the currently active axis
W	Set Axis "W" axis as the currently active axis

X	Set Axis “X” axis as the currently active axis
Y	Set Axis “Y” axis as the currently active axis
Alt+Z	Zeroes the specified axis, or all axes
Z	Set Axis “Z” axis as the currently active axis
<TAB>	Toggles between continuous and discrete jogging modes.
<ALT> + 1-8	Activates the selected axis. After an axis is activated, you may select a jog distance that the axis will be moved each time the associate jog keys are depressed
1-0	Selects an index into the active increment table for the currently active axis. If the axes are synchronized, all increments for the current unit table are updated. If the unit tables are synchronized, the appropriate entries in both tables are updated.
Left Arrow	Moves the tool along its associated axis in a negative direction. When TurboCNC controls a mill, this axis is usually the X axis.
Right Arrow	Moves the tool along its associated axis in a positive direction. When TurboCNC controls a mill, this axis is usually the X axis.
Up Arrow	Moves the tool along its associated axis in a positive direction. When TurboCNC controls a mill, this axis is usually the Y axis.
Down Arrow	Moves the tool along its associated axis in a negative direction. When TurboCNC controls a mill, this axis is usually the Y axis.
Page Up	Moves the tool along its associated axis in a positive direction. When TurboCNC controls a mill, this axis is usually the Z axis.
Page Down	Moves the tool along its associated axis in a negative direction. When TurboCNC controls a mill, this axis is usually the Z axis.
Plus (+)	Moves the tool along its associated axis in a positive direction. When TurboCNC controls a mill, this axis is usually the A axis, which is a rotary table.
Minus (-)	Moves the tool along its associated axis in a negative direction. When TurboCNC controls a mill, this axis is usually the A axis, which is a rotary table.
<ALT>	When jogging in continuous mode, selects the high jog rate for the axis. When in discrete mode, has no effect on jog speed.

MDI Mode

Selects the Manual Data Input mode of operation. RS-274 D code can be entered and immediately executed. Instructions that alter the order of execution such as subroutine calls (M98) and returns (M99) are not allowed.

The MDI mode is a convenience that allows you to input single blocks of G-Code and have them executed immediately, without executing a file. This is useful for moving large distances, or to make practice cuts. For some simple parts, you might do all of the cutting in the MDI window instead of writing a program.

Everything input in MDI mode is copied to a file called MDI.CNC, along with a date/time stamp located in the same directory as the TurboCNC executable.

Home Axes

The speed at which TurboCNC homes the axes is configurable under the Configure/General menu.

Feed Override Adjust

Changes the feed override value from 10% to 1000%. When the feed override is active, an asterisk will be displayed in the Status window next to the feed rate, which will be adjusted to show the “true” feed rate that is in use. 100% is “no override”, that is, the feed rate will be the same as programmed.

Adjust the feed override downward to compensate for cutting conditions like dull tools, hard spots in materials, and poor clamping. Adjust it upward to compensate for melting or burning materials, chatter, or for faster production if conditions otherwise allow.

Toggle Show Backlash

Shows or hides the backlash direction indicators in the status display for all axes that have a backlash configured.

Toggle Optional Breaks (M01)

Enables or disables M01 Optional Breaks depending on its current state.

Toggle Marked Blocks (‘/’)

Enables or disables the block delete mode depending on its current state.

When enabled TurboCNC will ignore blocks of code that have this as the first valid character on a line. The block delete character may be preceded by 'white space' such as space or tab characters and comments enclosed in brackets.

If this is enabled and the block delete character is encountered after the first word in a block, TurboCNC will ignore the following word only.

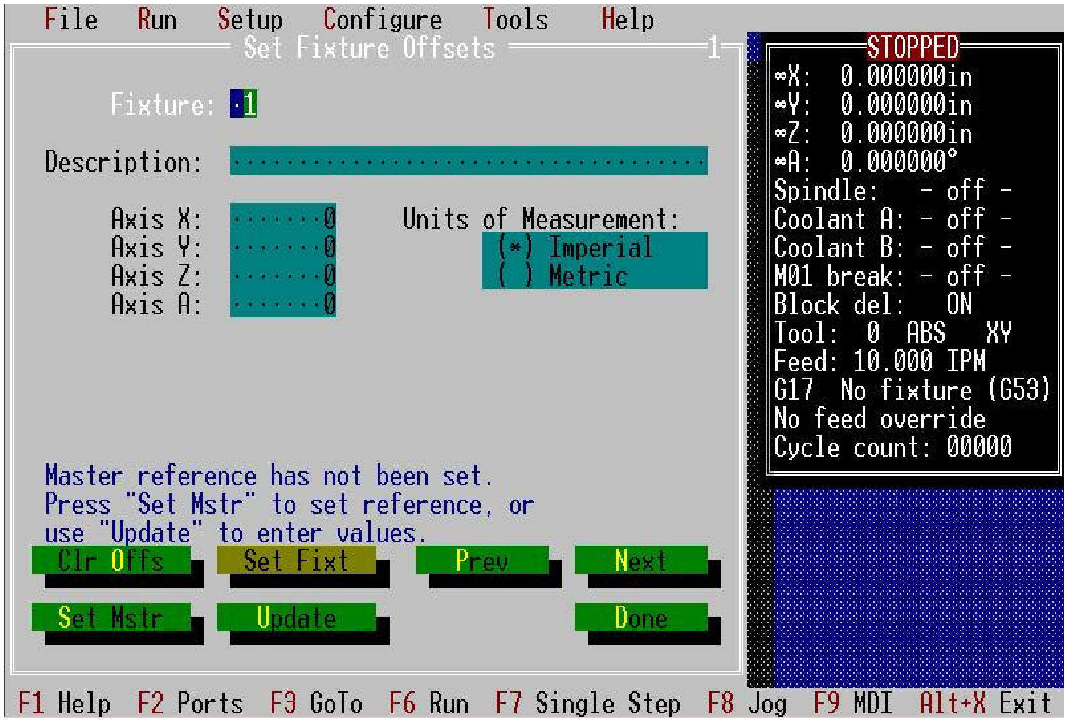
```
G00 X0 /Y0 Z0     ; y-axis does not move if block delete is on
/G00 X0 Y0 Z0     ; no movement if block delete is on
(*** no movement if block delete is on ***) /G00 X0 Y0 Z0
```

Toggle Working Units

Switches the system's working units of measurement from Imperial to Metric or vice versa. This menu item has the same effect as G20/21/70/71. See the 'Switch Native Units to ' section under Configuration for a discussion of working and native units of measurement.

Manage Fixtures

The "master coordinate" system must first be set if jog is to be used in setting the offsets. If the machine has home switches, it may simply be "homed". If the machine does not have home switches installed, it can be jogged to a designated "home" position. The master coordinates can be set as desired at the "home" location, athough this is normally the point of origin. All fixture offsets are caculated with respect to this location.



A confirmation dialog box will appear ensuring that fixture offsets are not inadvertently cleared. Following the operation a message box will appear to confirm that the fixture offsets have been successfully cleared.

Manage Tooling

A means of setting tool offsets is provided under this menu item.

Before any of the tooling offsets can be set, a reference location to zero must be set. Once the reference is set to a zero location, all other tools are to be moved to this location. You may use either a tool as the reference, or may make a special tool setting gauge. Once the reference location has been set, load the actual tools and move them to the reference location, using the jogging controls. TurboCNC will then record the offset locations for each axis.

- Select Setup->Manage Tooling from the main menu
- Load the reference gauge in the spindle.
- Move the reference gauge to a reference location using the cursor to jog the reference into place. Either build a gauge to provide a point to which the reference gauge is moved, or use some portion. Allow the motors to move the reference. If power to the motors is cut and the axes manually moved, TurboCNC will not be able to record the position, and will not accurately set any of the tooling offsets.
- With the reference gauge in the correct location, press the "Set" button to record the reference coordinates.

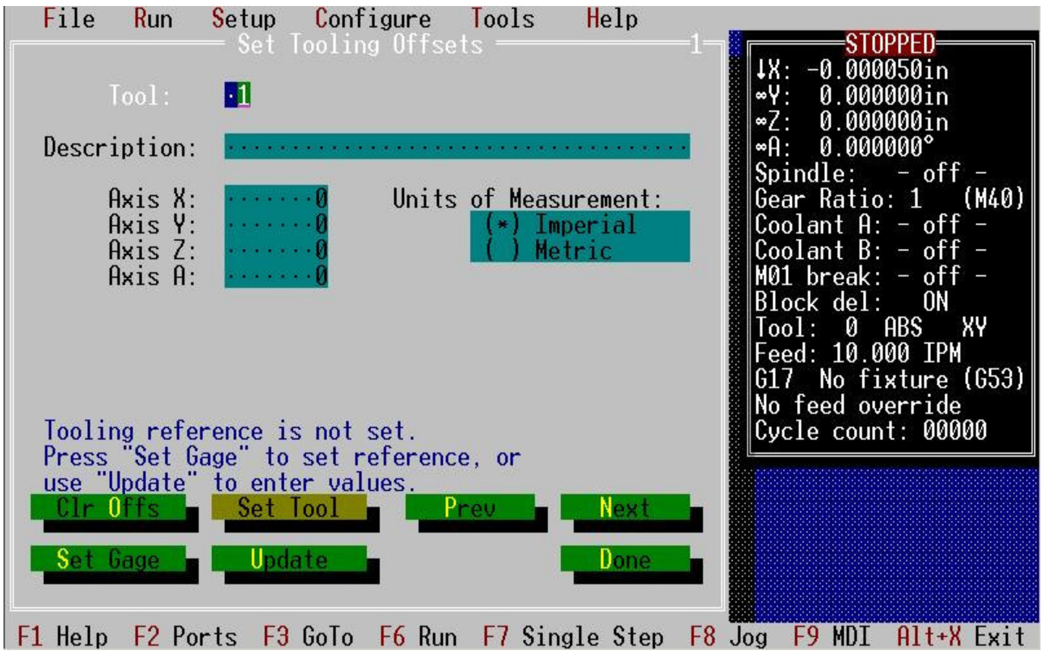
Now that the reference location has been set, TurboCNC shows a screen allowing the offsets for each of the tools to be set. This is done as follows:

- Use the "prev" and "next" buttons to select the tool for which the offsets are to be entered.

- Enter a description of the tool on the line provided, if desired. This description is not necessary, but can be used to identify the tool during execution of the CNC program.
- Load the tool into the spindle and jog it to the same reference point to which the gauge was set.
- Press the "S" key to set accept the location and calculate the tool reference, or press the "C" key to cancel the function and revert to the current tool offsets.
- TurboCNC will store the offsets in a table, and will automatically select the next tool in series.
- Tools may be directly edited from this menu.
- Rear Toolpost is not yet active, and has been included for future development.

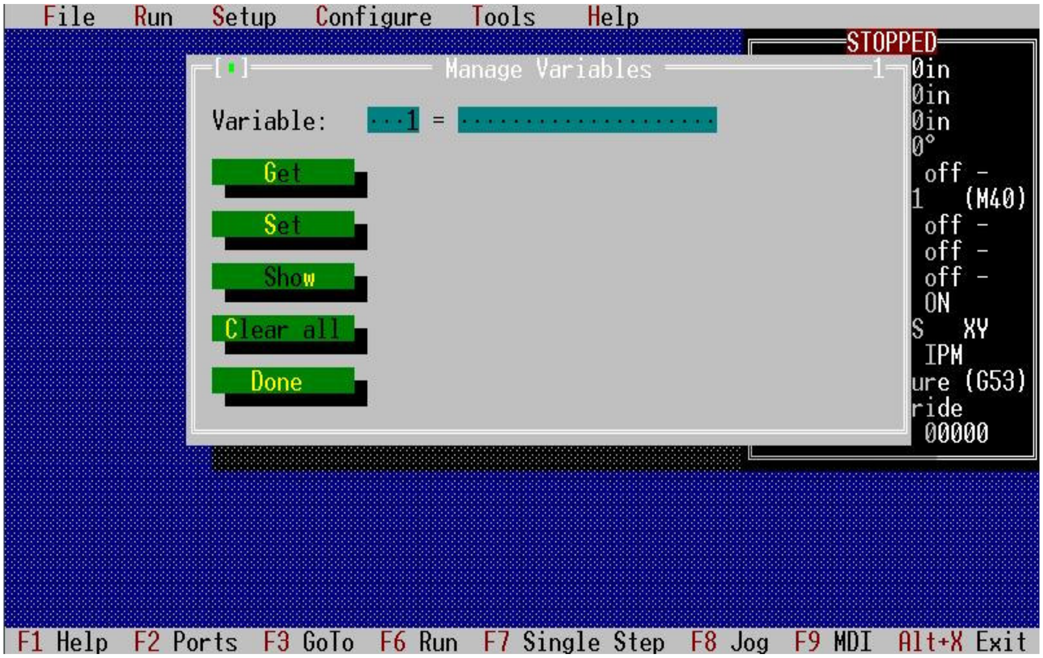
When all of the tool offsets have been set, press the "done" button to exit the function. An opportunity to save the new tool offsets will be presented. The offsets will be stored in a file located in the directory as specified in the Configure->General dialog.

The screenshot below shows the options prior to 'Set Gage' .



A confirmation dialog box will appear ensuring that tool offsets are not inadvertently cleared. Following the operation a message box will appear to confirm that the tool offsets have been successfully cleared.

Manage Variables:



Provides a means of setting, and inspecting variables without using RS-274D in MDI or a custom program. ***Show*** will bring up a scrolling list of all variables that are currently storing a value. ***Clear all*** will reset all variables to an empty string.

Disable Drives

This option is available only if a drive enable line has been configured. All drive enable lines are set to their inactive states when this option is selected.

Enable Drives

This option is available only if a drive enable line has been configured. All drive enable lines are set to their inactive states when this option is selected.

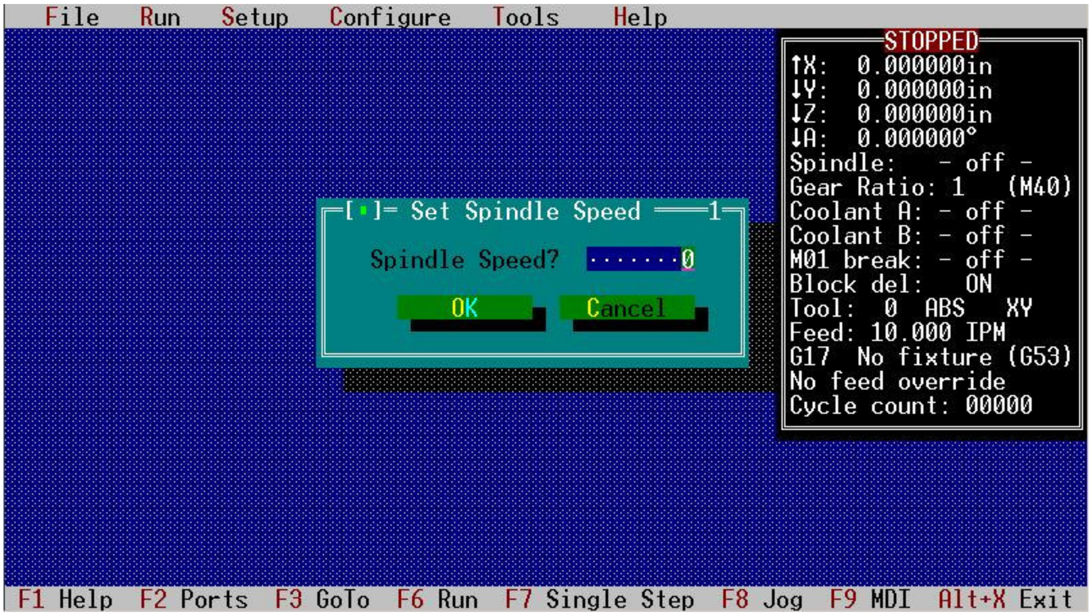
Reset DigiSpeed Control

Commands are sent to the spindle speed controller, disabling it, and resetting the speed to 0 RPM.

Set Spindle Speed

A dialog box is presented allowing the user to directly enteer the desired spindle speed. The command is sent to the speed control.

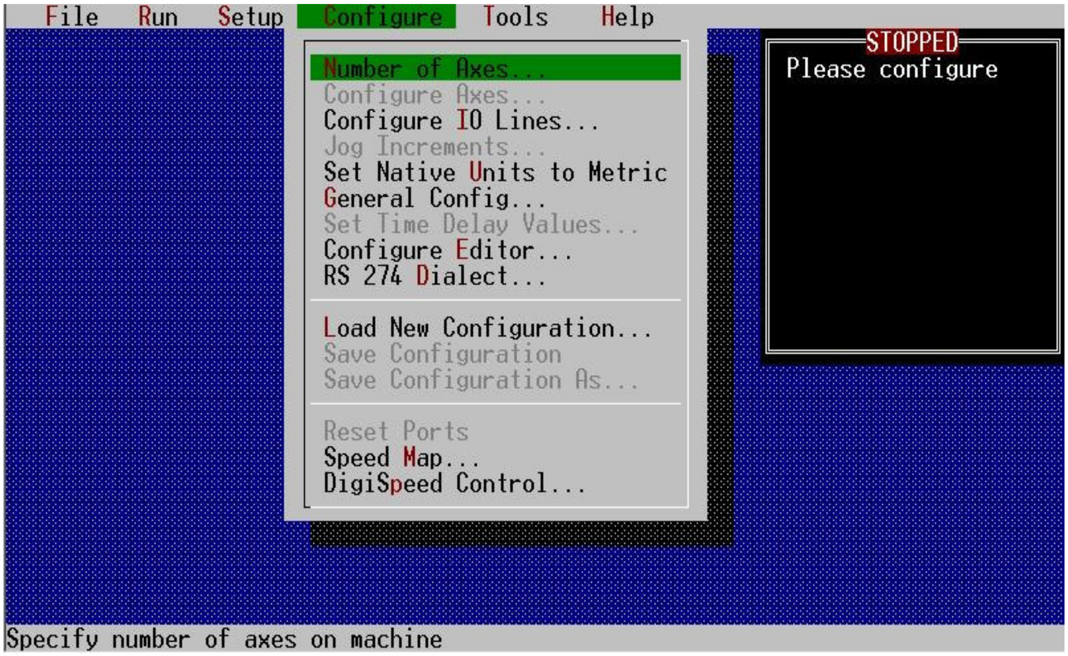
NOTE: neither of these options affect the state of the Spindle Power control line. This must be set separately.



Configure

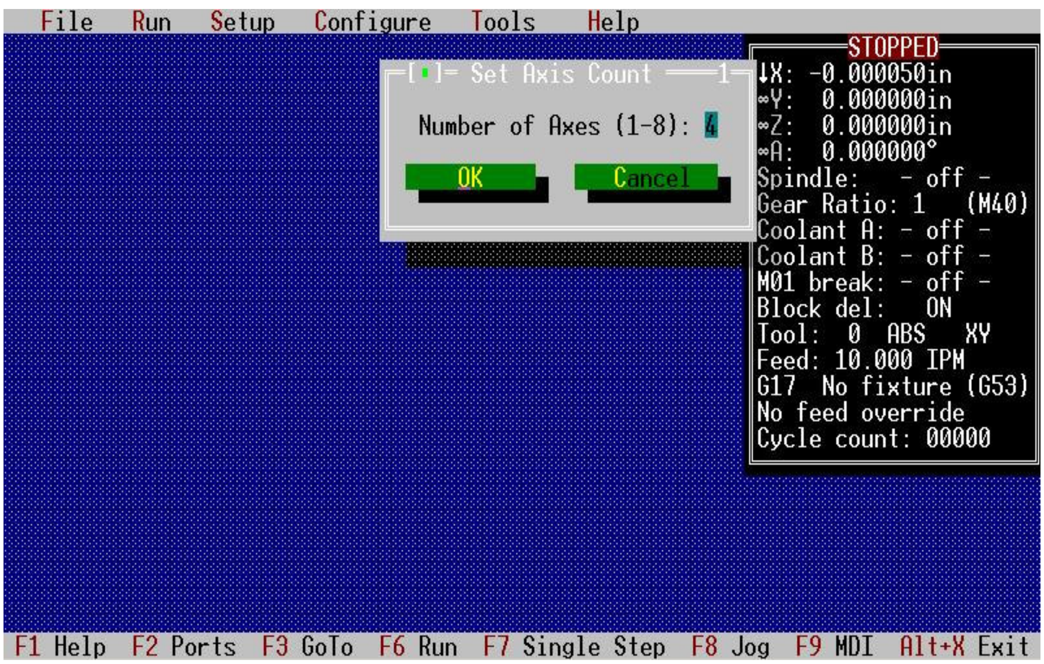
The 'Configure' option on the menu bar is used to setup the parallel port I/O to drive your system, configure the way TurboCNC interprets code, set tool and fixture offsets, and maintain the configuration file (turbocnc.ini). The screen below shows the configuration menu for TurboCNC when it is started without a configuration file. Several menu items are grayed out indicating that they not available until the number of axes has been configured. These are:

- Configure Axis
- Jog Increments
- Set Time Delay Values
- Save Configuration
- Save Configuration As, and
- Reset Ports



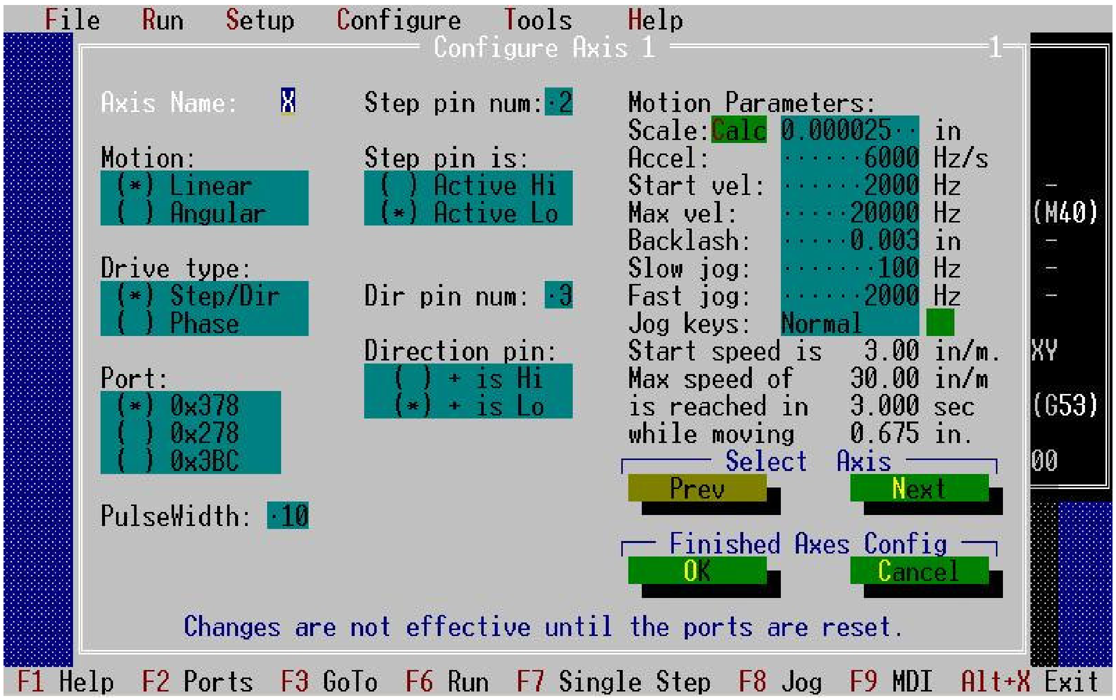
Number of Axes…

Allows specification of the number of axes on the machine that TurboCNC will drive. If no axes are specified TurboCNC will not save the configuration file on exit. The number of axes on the machine cannot be changed while a parts file is open.



Configure Axis…

is used to set the parallel port I/O, and specify the motion parameters for each axis. Upon selection, a dialog box is presented where the first axis to be configured can be selected. You will then be taken to the main configuration screen shown below. This screen is dynamic and will change to reflect your choices. It is recommended that you use the mouse to navigate this screen, as the design trade off for the dynamics was intuitive keyboard navigation.



Items on the axis configuration menu are:

Axis Name: Assigns a drive letter that is used to select this axis in RS-274 D. Valid selections are A through E and U through Z.

Motion: selects whether the axis is linear or angular. Angular axis measurements are always in decimal degrees, modulus 360, and driven the shortest distance to their new position. Linear axes are measured in either inches or millimeters, depending on the system of measurement in use.

Drive type: selects whether a Step/Direction or Phase scheme is used to control the motor driver. A Step/Direction scheme requires only two output pins, while a phase scheme requires a minimum of 4, and up to eight output pins to drive a single motor. Drive type selection determines whether Step/Direction or Phase definition configuration information is displayed on the menu.

Pulse Width: changes the duration of the step pulse on step/dir controlled axes, as some drives need a few microseconds to recognize that the step line has changed state. The parameter is set directly in integer microseconds. 0 is no explicit delay, which works out to be between about 2 and 7 µs on most computers.

Port: The parallel port to which the driver is connected is selected by this option.

Step pin num: sets the output pin on the selected port for step pulses. Valid values are 2, 3, 4, 5, 6, 7, 8, 9, 1, 14, 16, and 17.

Step pin is: This option is used match TurboCNC to your controller's requirements. Check you controller documentation, or the files section of the TurboCNC newsgroup on Yahoo! to determine the proper setting. (Trial and Terror can also be used.)

Dir pin num: sets the output pin on the selected port for the direction signal. Valid values are 2, 3, 4, 5, 6, 7, 8, 9, 1, 14, 16, and 17.

Direction pin: selects the polarity required to drive the axis in the positive direction. Phase wiring of the motor and driver electronics determine this setting. Jog or manually move the axis to a position where you can safely move the axis and use relative movements in MDI to determine if the axis moves in the proper direction. If not, simply toggle this selection and verify that the axis now moves in the correct direction, once again using the MDI mode of operation.

Motion Parameters: are used to set various values governing the movement of the axis. Units of measurement will vary to reflect the type of axis and units of measurement in use by TurboCNC. Acceleration, velocity, and scale effects are reflected below the data entry area. There is no one-size-fits-all solution. These setting depend on all of the equipment, and the cutting forces to be used. A starting point for several systems can be found in the files area of the TurboCNC Yahoo! Group or in the inifiles subdirectory of your installation. These settings will have to be fine tuned to maximize the system's performance.

Scale: is the distance or angle that the axis moves for a single step pulse or phase change. The units can be in inches, millimeters or decimal degrees, depending on axis motion, and the system of measurement in use. A custom calculator is accessable by clicking on 'Calc, or pressing 'c'.

Accel: the maximum acceleration of the step pulse train or phases that are sent to the driver measured in cycles per second per second. This is purely electronic, and is converted to motion by the motor. ***Scale*** is used to convert this value to the linear or angular value displayed below.

Start vel: is the maximum starting / stopping velocity that will be used by TurboCNC for this axis. Lower values will often be used for interpolated moves. This is measured in cycles per second, and converted to a distance or angular measurement using ***Scale*** and displayed below as the ***Start Speed***.

Max vel: is the maximum velocity that TurboCNC will drive the axis. This is measured in cycles per second, and converted to a distance or angular measurement using ***Scale*** and displayed below as the ***Max Speed***.

Backlash: is the compensation applied whenever the axis changes its direction of movement. It is measured in inches, millimeters, or degrees depending on the type of motion and system of measurement in use.

Slow jog: This is the speed at which the step pulses are applied, or phase changes occur when ***Slow jog*** is engaged. It is measured in cycles per second. The value can be converted to a distance or angular measurement by multiplying it by the ***Scale***.

Fast jog: Similar to ***Slow jog***, but allows you to select a higher speed.

Jog Keys: Selection of normal or invert is possible. In the inverted mode the motion of the tool with respect to the work piece is reversed for the axis.

Select Axis buttons accept the changes to the current axis and allow navigation through the axes to configure them without returning to the main menu and drilling down through the menu system to access the next axis.

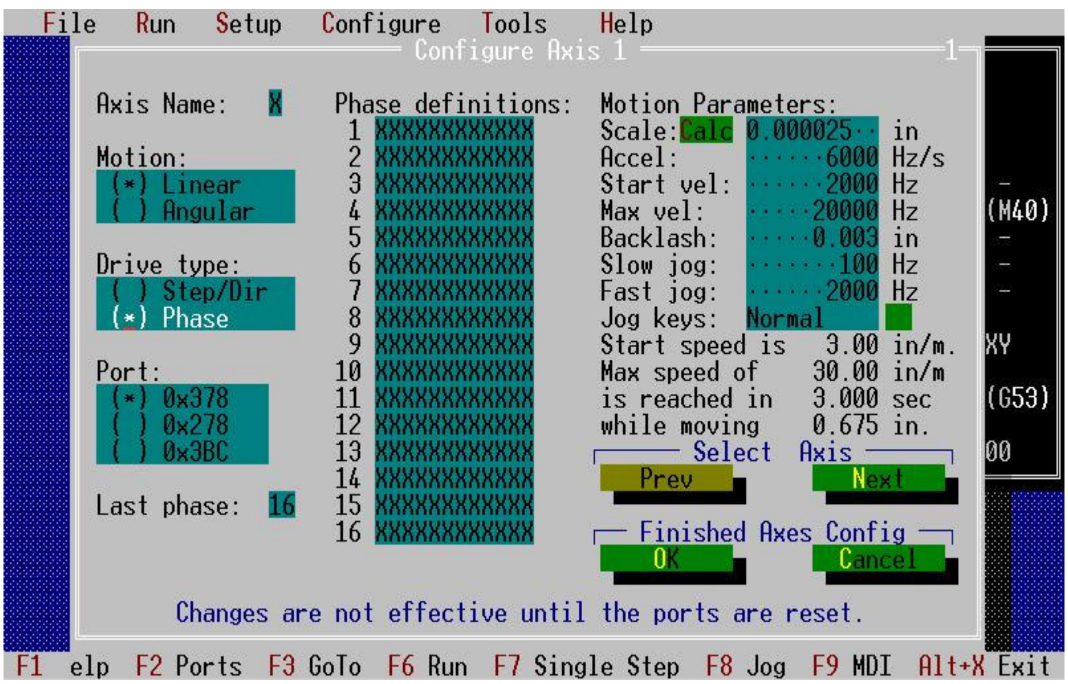
Finished Axis Config buttons are used to exit the axis configuration menu. '***OK***' accepts the changes, '***Cancel***' exits without saving the changes. **NOTE:** The ports must be reset to enable the new configuration. The configuration must be saved for it to be available the next time that TurboCNC is loaded.

The following screenshot shows the axis configuration menu for a phase driven, linear axis. Note that phase definition information has replaced the Step and Direction pin configuration parameters.

Set the ***Last Phase*** to the number of phases used in your drive scheme. Full, half, and quarter step schemes can be developed for both unipolar and bipolar two-phase stepper motors using the 16 phase definitions available. Pin assignments for ***Phase definitions*** follow.

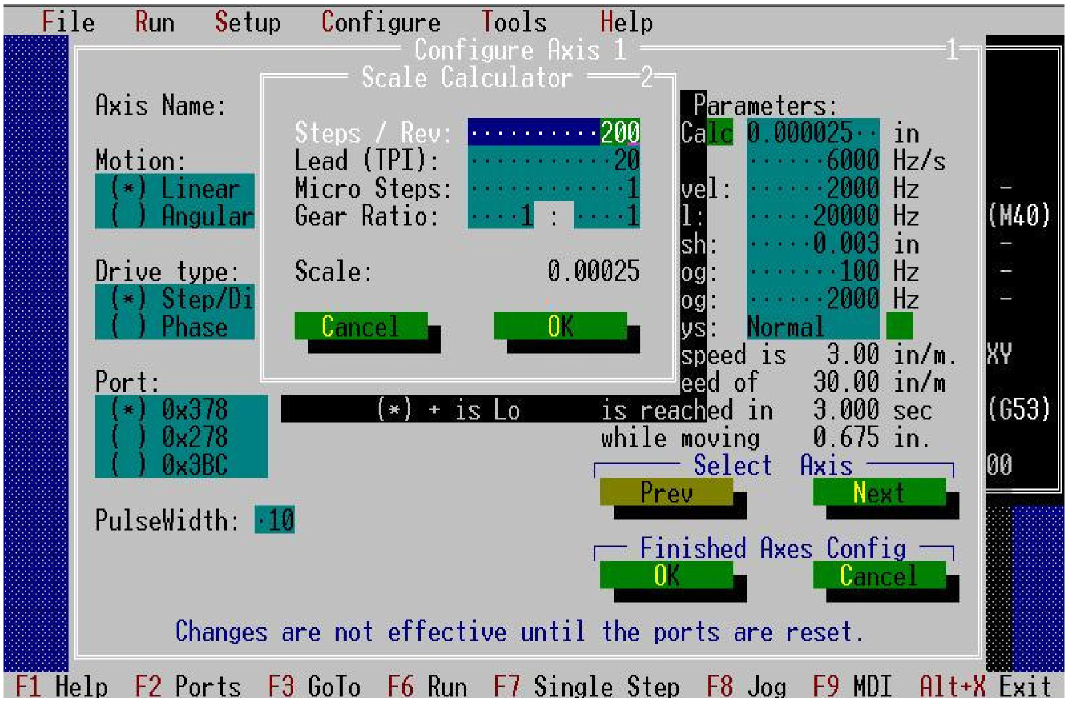
```
Pin state: 1 0 0 0 X X X X X X  X  X   (from Phase definition 1)
Pin ID:    2 3 4 5 6 7 8 9 1 14 16 17
```

Pins 1,14,16,17 are driven separately from the other 8 since the computer can only change one byte at a time on the port. Therefore, keep all the phases for one motor either on pins 2-9 or on pins 1,14,16 and 17.



Scale Calculator

A custom Scale calculator is available from the Axis Configuration Menu. This is used to compute Scale using common factors. In the Imperial system of measurement the TPI of the lead screw is used, while in the Metric system the pitch is used. Values entered into the scale calculator are persistent through the current session, and are not saved to the configuration file. The scale displayed on the calculator is automatically updated when changing fields. Selecting OK will enter the computed value into the Scale variable on the current axis configuration menu. Cancel or 'Esc' will discard the computed value. These selections will close the calculator and return to the Axis Configuration Menu.



Configure I/O Lines…

There are 40 functions, which can be mapped to parallel port pins if you CNC machine supports the associated features available within TurboCNC. It is not necessary to map all of these functions. Just map the ones that you will use. The available functions are described elsewhere in this manual, and are listed below:

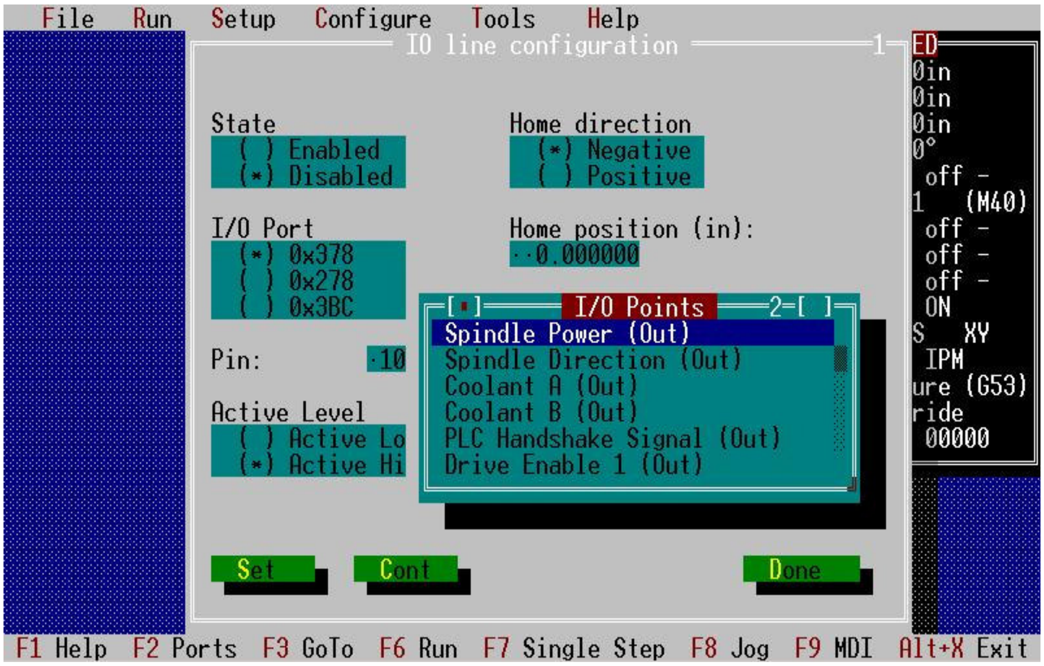
Number	Description	Direction	Number	Description	Direction
1	Spindle Power	Out	21	Limit Switch 3	In
2	Spindle Direction	Out	22	PLC Handshake Sense	In
3	Coolant A	Out	23	Spindle Index	In
4	Coolant B	Out	24	Spindle Encoder A	In
5	PLC Handshake Signal	Out	25	Spindle Encoder B	In
6	Drive Enable 1	Out	26	Touch Probe	In
7	Drive Enable 2	Out	27	Jog Encoder A	In
8	Drive Enable 3	Out	28	Jog Encoder B	In
9	Clamp Selector Bit 0	Out	29	Block Hold	In
10	Clamp Selector Bit 1	Out	30	Start Inhibit	In
11	Clamp Selector Bit 2	Out	31	Clamp Sense Opened	In
12	Clamp Selector Bit 3	Out	32	Clamp Sense Closed	In
13	Clamp Motor On Signal	Out	33	Home Switch 1	In
14	Clamp Direction Closed	Out	34	Home Switch 2	In
15	Tool Turret Index	Out	35	Home Switch 3	In
16	Collet Open Solenoid	Out	36	Home Switch 4	In
17	Collet Close Solenoid	Out	37	Home Switch 5	In
18	Emergency Stop	In	38	Home Switch 6	In
19	Limit Switch 1	In	39	Home Switch 7	In
20	Limit Switch 2	In	40	Home Switch 8	In

Each function allows you to enable it and configure the parallel port, the I/O line, and active logic level it will use. In addition the Home Switch functions allow you to specify whether the switch is located at the positive or negative end of the associated axis' travel, and what the position should be set to upon tripping the switch.

Select an item from the list box to configure it. Click on 'Set', or enter alt + S from the keyboard to save the configuration and set another item. 'Cont' will bring up the selector list box without saving the change. 'Done' will save any changes and exit the 'IO Line Configuration' menu.

Description	Activate	Deactivate	Description	Activate	Deactivate
Spindle Power	M03 – CW M04 - CCW	M05	Limit Switch 3		In
Spindle Direction	M03 – CW M04 - CCW		PLC Handshake Sense		In
Coolant A	M07	M09	Spindle Index		In
Coolant B	M08	M09	Spindle Encoder A		In
PLC Handshake Signal	M70 M71	Out	Spindle Encoder B		In
Drive Enable 1	M17	M18	Touch Probe		In
Drive Enable 2	M17	M18	Jog Encoder A		In
Drive Enable 3	M17	M18	Jog Encoder B		In
Clamp Selector Bit 0		Out	Block Hold		In
Clamp Selector Bit 1		Out	Start Inhibit		In
Clamp Selector Bit 2		Out	Clamp Sense Opened		In
Clamp Selector Bit 3		Out	Clamp Sense Closed		In
Clamp Motor On Signal		Out	Home Switch 1		In
Clamp Direction Closed		Out	Home Switch 2		In
Tool Turret Index		Out	Home Switch 3		In

Collet Open Solenoid	Out	Home Switch 4	In
Collet Close Solenoid	Out	Home Switch 5	In
Emergency Stop	In	Home Switch 6	In
Limit Switch 1	In	Home Switch 7	In
Limit Switch 2	In	Home Switch 8	In



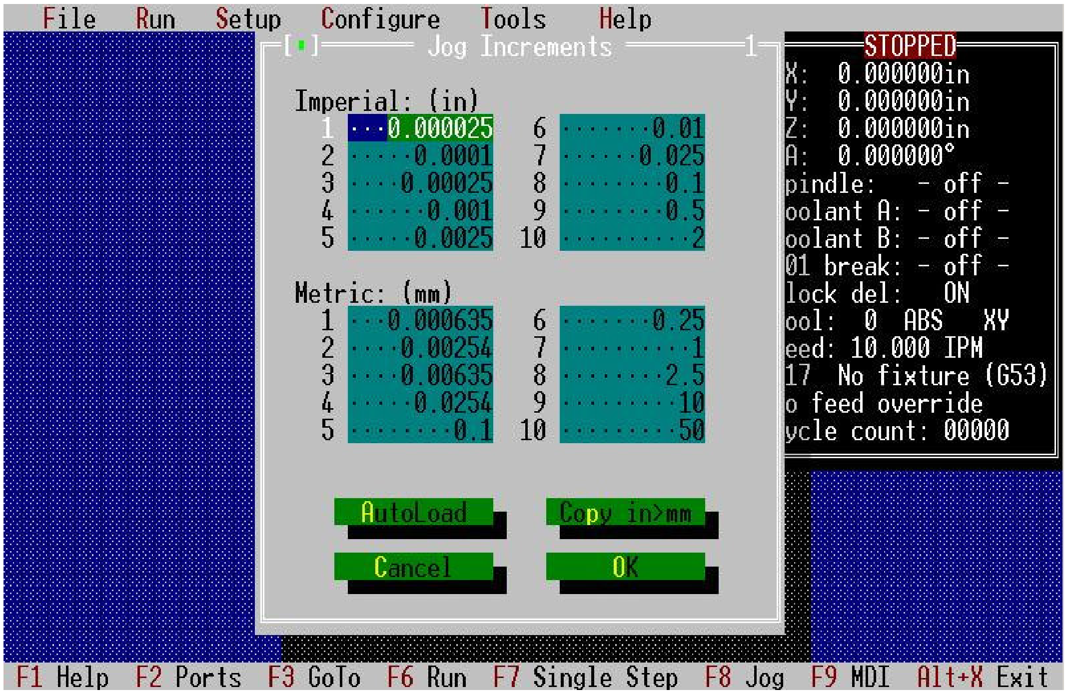
Jog Increments

This is an adaptive menu, meaning that the display is dependant upon the current configuration. The blocks for Imperial and Metric increments are exchanged when Metric is Native.

AutoLoad: The block for the Native system of measurement (always on top) is loaded with values based on the ***minimum*** step size of the configured axes. The first value is the minimum step size, the second is four times this step size. Subsequent values are ten times that of the value two increments below the current increment.

Copy in>mm: This is the label when the Imperial system of measurement is Native. The label is **Copy mm>in** when Metric. Selecting this option converts, and copies the increment values for the Native system of measurement to the non-Native system.

Hint: Use AutoLoad to generate the increments for the native system of measurement, followed by Copy, to load the the remainder of the increments. Finally, manually tune the larger increments so that they present a more natural fit to the selected system of measurement and the machine being controlled. The screen shot below shows the modified increments for 0.000025" minimum step size, on a mill with 9" X 7" X 5.5" travel.



Change Native Units to Metric (Imperial)

This menu choice changes dependent upon the system's current setting. It's recommended that this setting be chosen based upon the system of measurement of the leadscrews fitted to the linear axes of the machine. A confirmation dialog box will appear to ensure that this setting is not accidentally changed. All internal items stored in the 'Native' system of measurement are converted into the selected system. These items are:

- Axes' current position (if linear)
- Axes' Home position (if linear)
- Axes' Scale (if linear)
- Axes' Backlash (if linear)
- Current Feed Rate
- Default Feed Rate
- Home Feed Rate
- All Canned Cycle Parameters

TurboCNC and Systems of Measurement

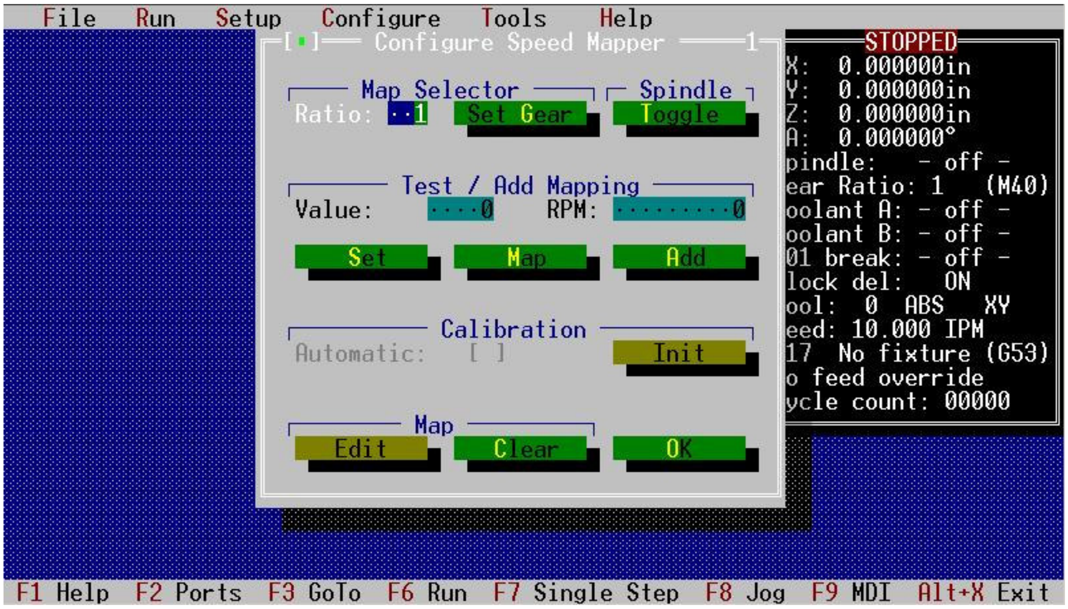
The TurboCNC core motion routines are no longer tied to any system of measurement. All positions for the linear axes are converted to the 'Native' system of measurement. When the calculations are performed, the units cancel out and we are left with a number representing the number of steps that must be taken, and the direction in which they are to be taken. This is referred to as the 'Unitless Motion Engine'.

There are now two systems of measurement that must be defined for your machine. The first is it's 'Native' system of measurement defined under the configuration menu. That is the only place Native units of measurement can be changed. This system should be chosen based upon the leadscrews used on the linear axes. All configuration items remain in this system of measurement. This eliminates the display idiosyncrasies of TurboCNC V4.00, and the conversion errors that occurred at the 13th decimal place.

The second is termed the 'Working' system of measurement. All linear axis positions entered by RS-274D are considered to be in this system of measurement. These values can be part of a program or entered through the MDI, or jog interfaces. The 'Working' system of measurement can be changed using G20/21/70/71 or the 'Toggle Working Units' item under the Setup Menu.

SpeedMap

Selection of SpeedMap displays a dialog box allowing up to four independent Speed Maps to be configured. Maps are selectable from RS-274D code using the M40 through M43 codes. The standard defines these codes as those that should be used for gear changes. As each speed map is intended to represent a gear ratio, these were used.



Ratio:

Used to select a Speed Map or Gear Ratio. Valid values are from 1 to 4. The number 1 corresponds to M40, 4 to M43.

Set Gear

This button is used to set the current speed map to that specified in the Ratio box.

Toggle

Spindle Toggle turns the spindle on or off depending on its current state.

Value

Used to enter a count value for use with the Set and Add buttons described below.

RPM

The speed in RPM is entered in this box for use by the Map and Add buttons described below.

Set

Sets the speed by Value, and turns the spindle on in the clockwise direction if it was off. This is used to determine the RPM associated with a count Value. The RPM corresponding to this value can be entered into the RPM box, and added to the current map by pressing Add.

Map

Map is used to retrieve the count associated with an RPM from the currently selected speed map. RPM values above or below the map limits will be reported as an error.

Add

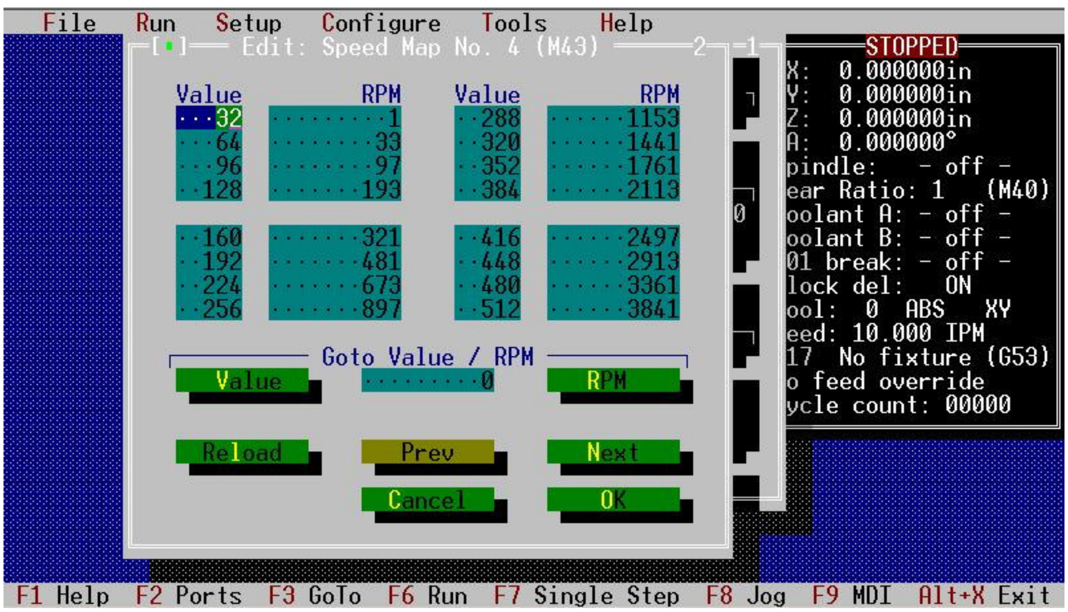
Used to enter the specified Value and RPM as a mapping in the currently selected map. The high and low RPM limits for the map are adjusted as required.

Calibration

These options have yet to be written.

Edit

Upon opening, the Map Editor sorts the items of the current Speed Map in ascending order, and validates the map. A local, working copy of the map items is made. If the map fails the validation check, 'Invalid Map' is displayed in the lower left coner of the form, and the display is indexed to the first offending item. If the validation check is passed, the display is indexed to the first map item.



Value

Moves the Edit Window so that the first count Value is is equal to the Target. If the Target is not found, the next lower count Value is used.

<u>**Note:**</u> The Target for both Value and RPM is entered into the box between these buttons.

RPM

Moves the Edit Window so that the first RPM is is equal to the Target. If the Target is not found, the next lower RPM in the speed map is used.

Reload

Reloads the local working copy of the Speed Map from the main map array. The Edit Window is indexed as when the editor first opened.

Prev

Moves the Edit Window 16 items down. If this is below the first item, it shifts the window to the first Item in the map.

Next

Moves the Edit Window up 16 items, or so that 16 blank entries are shown. Next has no effect if all blank items are shown. If this is above the last item, the last item in the Window will be the last item in the map.

Cancel

Exits the Map Editor without saving changes.

OK

Uploads the working copy of the map to the main map array. The map is then validated, and if it is valid exits the editor. If the map fails to validate, the editor will remain open and position the Edit Window to the first invalid item.

Note: If the map does not validate properly it cannot be used to set spindle speed. It must first be corrected within the Editor, or the original values can be reloaded from disk by cancelling out of the Map Editor, and reloading the configuration file.

Clear

Clears all values from the currently selected map, and sets the RPM limits to default values of 100,000,000 for the low end and 0 for the high end limits.

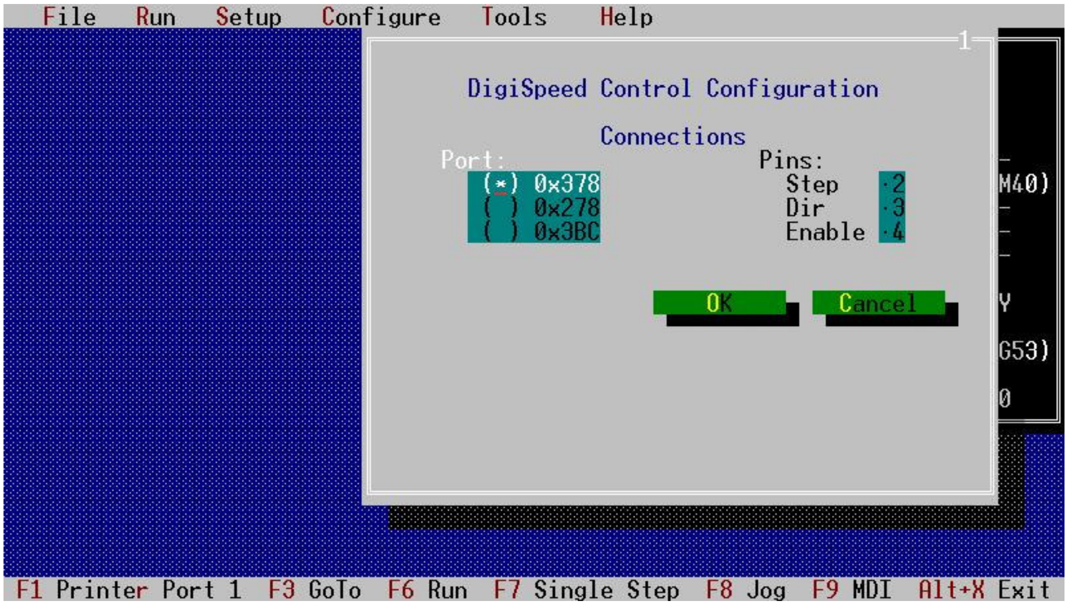
OK

Exits the menu, accepting all changes. To revert to a previous configuration, this must be re-loaded from disk. There is no provision to reject changes made in the menu.

NOTE: The updated configuration must be saved to preserve any changes between sessions of TurboCNC. Simply exiting TurboCNC, and re-starting it will erase any changes made.

DigiSpeed

Selection of the DigiSpeed Control displays a dialog box to configure the DigiSpeed Control.



Port

This selects the port that will be used for all communications with the DigiSpeed.

Pins:

Allows selection of the pins for Step, Direction and Enable signals to be specified for the DigiSpeed. The active state for these signals is hard coded within TurboCNC.

General Config…

General configuration items have been gathered on this menu.

Color Menus: Default is to use color menus. When de-selected a monochromatic monitor can be used. Display Mode cannot be changed while a parts file is open.

Verbose Messages: By default TurboCNC will ask for confirmation of many actions. Turning this option off will substantially reduce the number of confirmations required.

Clear MDI Block: Check this box if you want TurboCNC to clear the MDI block that you entered after it executes that block. Clear the box if you want the edit box to retain the block's contents after executing the block.

Mouse Off During Move: The cursor is hidden by default during motion to conserve CPU cycles. This can be turned off when TurboCNC is running on faster computers.

Home Switch is Limit. Enable this option if the Home Switches are to be used as Limit Switches. The Home Switches should be defined only as such. During Home moves TurboCNC de-couples these switches from their Limit function and uses them solely as Home switches. For all other moves they will function as limit switches.

Stop on Illegal G-Code: This option is used to provide the ability to execute G-Code programs containing instructions which are **NOT IMPLEMENTED** in TurboCNC. If you check the box, TurboCNC will stop execution upon encountering an unimplemented code and display an information box indicating the errant code. If you clear the box, TurboCNC will simply ignore the unimplemented G-Code and continue executing your CNC program.

WARNING !	Clearing the check box and allowing TurboCNC to ignore unimplemented codes can have unintended consequences **TIP:** Examine each program which requires you to ignore unimplemented codes and assure that you can safely ignore those unimplemented codes.

Start Inhibit: You may designate an I/O point using **Configure I/O lines** which is sensed before a CNC program is allowed to execute. You may either abort the CNC program, or wait until the line returns to its inactive state. The default is to wait until the line returns to its inactive state.

Block Inhibit: You may designate an I/O point using **Configure I/O lines** which is sensed before a block in the CNC program is allowed to execute. You may either abort the CNC program, or wait until the line returns to its inactive state. The default is to wait until the line returns to its inactive state.

Sync Unit Increments: Check this box to direct TurboCNC to synchronize the indices of the Imperial and Metric tables for the currently active axis. For example, assume that you are working with the Imperial jog table, and that the current index value of the Imperial table is 5 and that the current index value of the Metric table is 3. If this box is unchecked and you change the Imperial table index to 9, the Metric table index remains at 5. If the box is checked and you change the Imperial table index to 9, the Metric table index also becomes 9.

Note that this check box affects only the 1-0 keys and that the J and K keys continue to affect **ONLY** the currently active table so you still have a way to change the distance of **ONLY** the active axis without affecting any of the others.

Sync Axis Increments: Check this box to direct TurboCNC to synchronize the index of all axes on a given table to the value selected. Note that **ONLY** the indices into the active table are set. For example, if X is the active axis, Imperial measure is the active table, and you press the “3” key, only the X index will be changed to 3. If however, the Sync Axes box is checked, all axis indices into the Imperial table are set to 3.

Note that this check box affects only the 1-0 keys and that the J and K keys continue to affect **ONLY** the currently active table so you still have a way to change the distance of **ONLY** the active axis without affecting any of the others.

These boxes work either in concert, or individually, so checking both boxes will make the jog appear as there is only one index into both tables.

Autoload Tooling File: Default is not to load Fixture and Tool offsets when TurboCNC starts. Enabling this option will attempt to load a tooling file from the Directory specified by ***Tooloff Dir*** with the name TURBOCNC and filename extension specified by ***Tooloff Ext*** whenever TurboCNC is restarted.

Imperial Precision: sets the number of digits to the right of the decimal point to which the backlash and scale are rounded when the display is set to use Imperial units. Valid values are 0 to 9. A value of 0 will prevent any rounding. This is only used when changing the Native system of Measurement to Imperial.

Metric Precision: sets the number of digits to the right of the decimal point to which the backlash and scale are rounded when the display is set to use SI units. Valid values are 0 to 9. A value of 0 will prevent any rounding. This is only used when changing the Native system of Measurement to Metric.

Note: Selection of too coarse a precision can cause axis scale value rounding to zero. This is checked on entry to and exit from the axis configuration menu.

Default Feed Rate: This option provides a method of setting a default feed rate, to be used when TurboCNC is first started.

Home Speed: A custom homing speed can be set using this option.

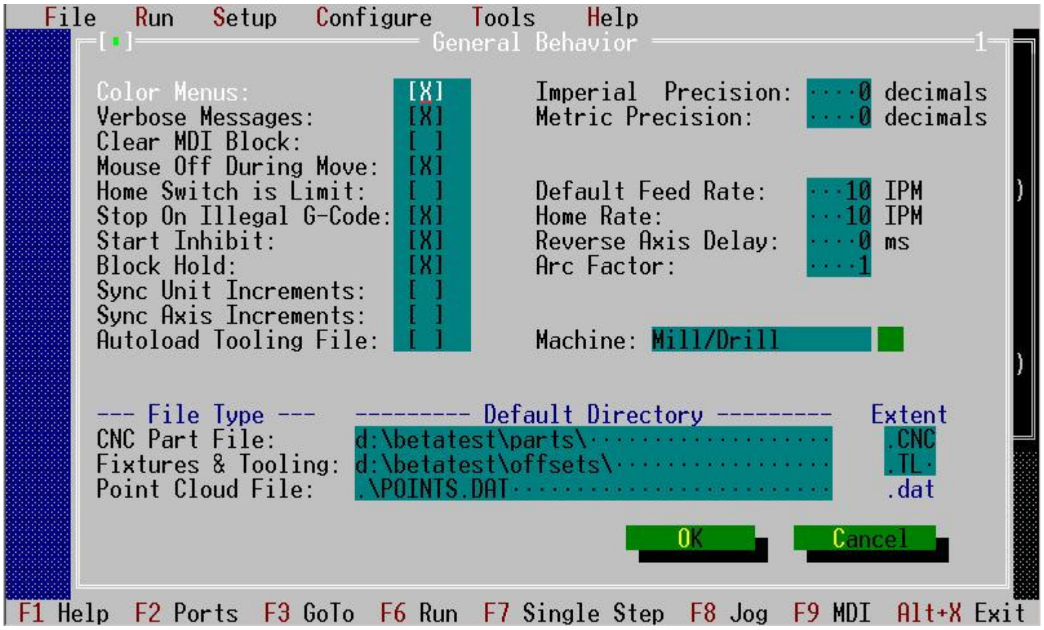
Reverse Axis Delay: specifies a delay in milliseconds that all axes will pause before changing direction. Axis delay is used to compensate for machine inertia. Whenever an axis reverses (backlash, arc quadrant, just plain reversal) the machine drives should actually pause for a short time to allow it come to a complete stop. The classic example is a big honkin' gantry machine with the moving mass 6' from the supports.

Arc Factor: This option provides a method of adjusting timing loops used when cutting an arc to the speed of the computer. Values less than one increase the loop speed, those above one decrease it. If you notice lost steps while cutting arcs decrease this value to 0.8 or less.

Machine: Allows be specification of the general type of CNC machine being controlled. This setting affects how TurboCNC interprets RS-274 D Code as follows:

- **Radius Lathe:** The X axis (cross feed) is used as specified when processing a move. This default plane is set to G18 at startup.
- **Diameter lathe:** The X axis (cross feed) is halved internally before processing a move. Earlier versions of TurboCNC had a cryptic way of setting this via an AxisPreScale parameter in the ini file. This default plane is set to G18 at startup.
- **Mill** / **Drill:** This default plane is set to G17 at startup.
- **Custom:** Has been defined for customers wishing to incorporate special setup procedures for their systems.

The remainder of the screen allows specification of the location and default extentions for CNC Part Files, Fixtures & Tooling files, and the Point Cloud File.



Set Time Delay Values …

Since a computer can often execute its instructions much faster than the world around it can react, TurboCNC often pauses after changing one of the I/O points that you might define. This menu entry allows you to tune TurboCNC's delays to match your hardware.

Relay Debounce: Specifies the time, in milliseconds, that TurboCNC waits for a mechanical relay to change its state. Codes affected: **M03, M04, M10, M11**

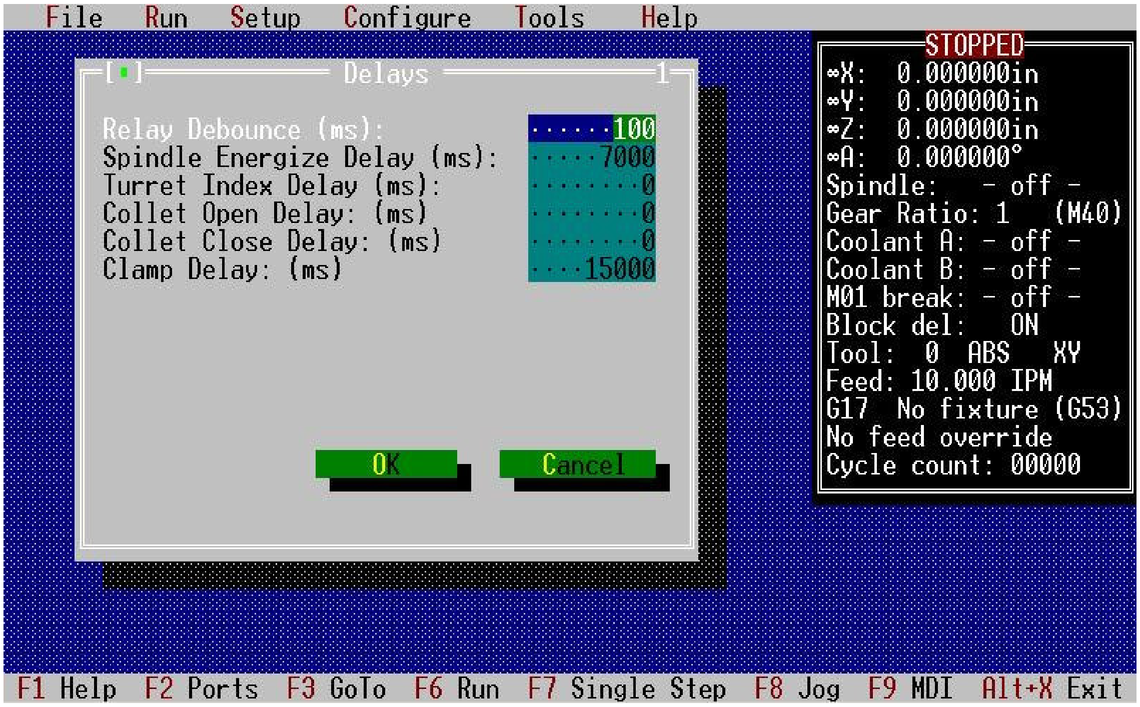
Spindle Energize Delay: Specifies the time, in milliseconds, that TurboCNC waits for the spindle to come up to speed or to slow down. Codes affected: **M03, M04**

Turret Index Delay: Specifies the time, in milliseconds, that TurboCNC waits to allow a tool changer to index to the correct tool. Codes affected: **M06**

Collet Open Delay: Specifies the time, in milliseconds, that TurboCNC waits to allow the collet to open. Codes affected: **M21**

Collet Close Delay: Specifies the time, in milliseconds, that TurboCNC waits toallow the collet to close. Codes affected: **M22**

Clamp Delay: Specifies the time, in milliseconds, that TurboCNC waits for a material clamping mecanism to engage or disengage. Codes affected: **M10, M11**



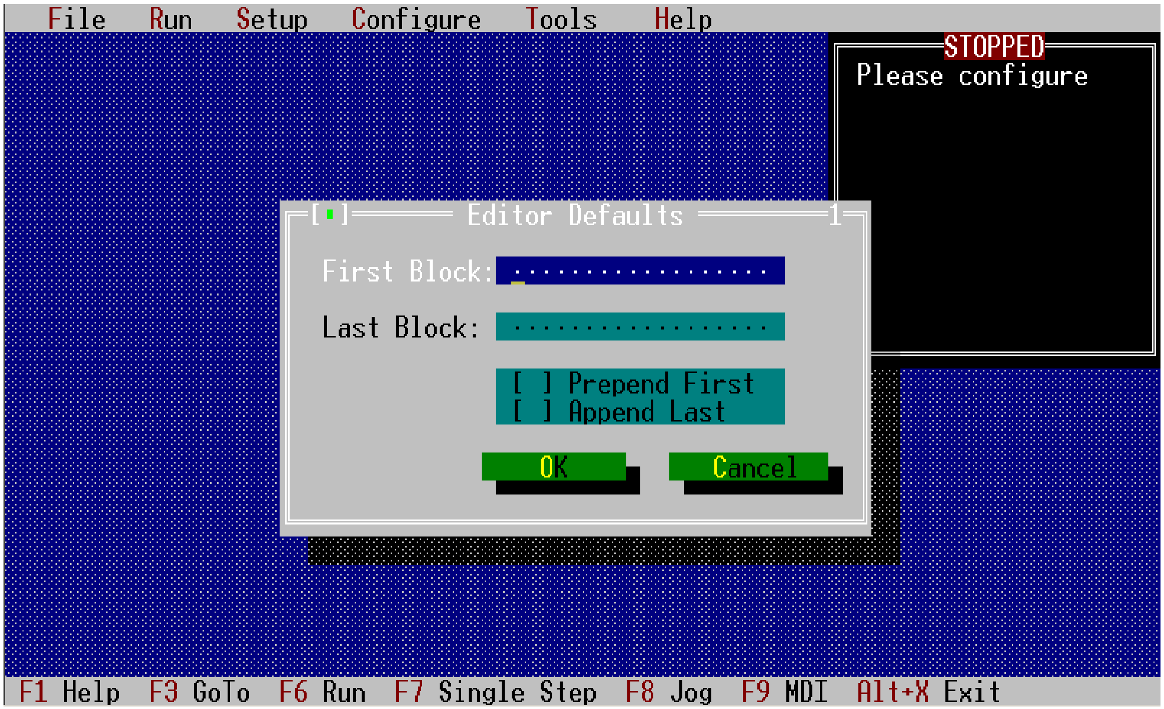
Configure Editor…

The editor can be configured to automatically add a block to the beginning and/or end of your CNC part files.

First Block: An optional 'First Block' that will be included whenever a new file is created using the 'New' choice of the 'File' menu.

Last Block: This mandatory entry will be included whenever a new file is created using the 'New' choice of the 'File' menu. The block will also be executed if the End-Of-File is reached before encountering a termination instruction (M02, M30) while running a file from the editor.

Existing File Options: When checked, the First and Last blocks specified above will be added to the beginning and / or end of a file loaded by selecting 'Open in Editor' under the 'File' menu.



RS 274 Dialect…

"The great thing about standards is that there are so many to choose from!"

TurboCNC allows some customization of its RS-274 dialect so that variations in programming styles can be bridged. The best practice is to use a custom post-processor in your CAD/CAM program for each controller that you use. If this is not possible, use these customizations instead.

G00 Linear Rapids: By default this is on, and all rapids are linear motions with each axis starting and stopping in unison. When it is off, each axis finishes moving as fast as possible. Overall, it's equally fast each way. Some people use the "dog-legging" that comes with a non-linear move to get around clamps and such.

G04 Dwells in ms: By default this is off. If you need a more precise delay, or if your CAM program assumes that the delay for G04 is in milliseconds, turn it on.

G82, 83, 183 Dwells in ms: By default this is on. If your CAM program assumes that the delays for G82, 83, 183 are specified in seconds, turn it off.

G33 Programmed as lead: By default this is on. If you prefer to program G33 in pitch as opposed to lead, turn it off. Note that in metric mode the lead and the pitch are the same.

M06 Jog Updates Location: This is off by default. Turn this on if you wish to update axis position while jogging during M06 Tool Changes.

M30 rewinds the program: By default this is off. If you want M30 to not only stop, but rewind the program as well, turn this on.

G04 Dwell Letter: Default is "P". You can set this to any letter except G, M, T, F or S.

G8x Dwell Letter: Default is "P". You can set this to any letter except G, M, T, F or S.

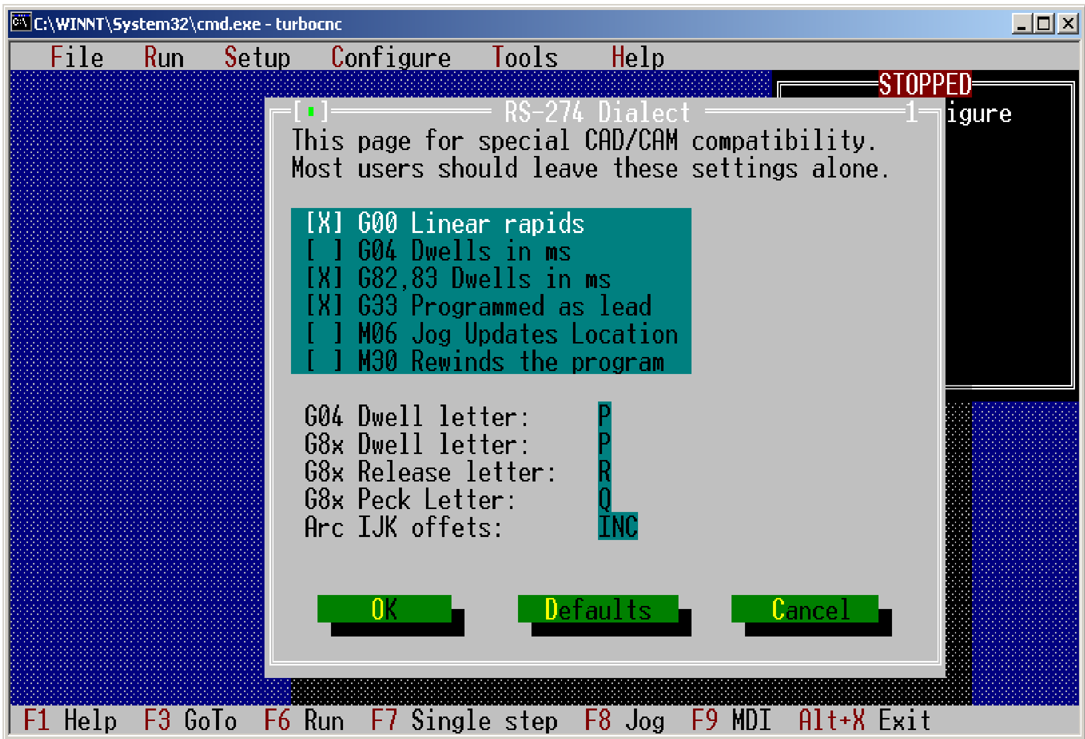
G8x Release Letter: Default is "R". You can set this to any letter except G, M, T, F or S.

G8x Peck Letter: Default is "Q". You can set this to any letter except G, M, T, F or S.

Arc IJK Offsets: Default is "***INC***", which stands for Incremental. There are two other settings, which govern the way that the IJK letters are interpreted:

> "***ABS***": IJK will always be interpreted as Absolute
>
> "***FOL***": IJK will follow the current mode. In absolute mode they are absolute values, in incremental mode they will be interpreted as incremental offsets.

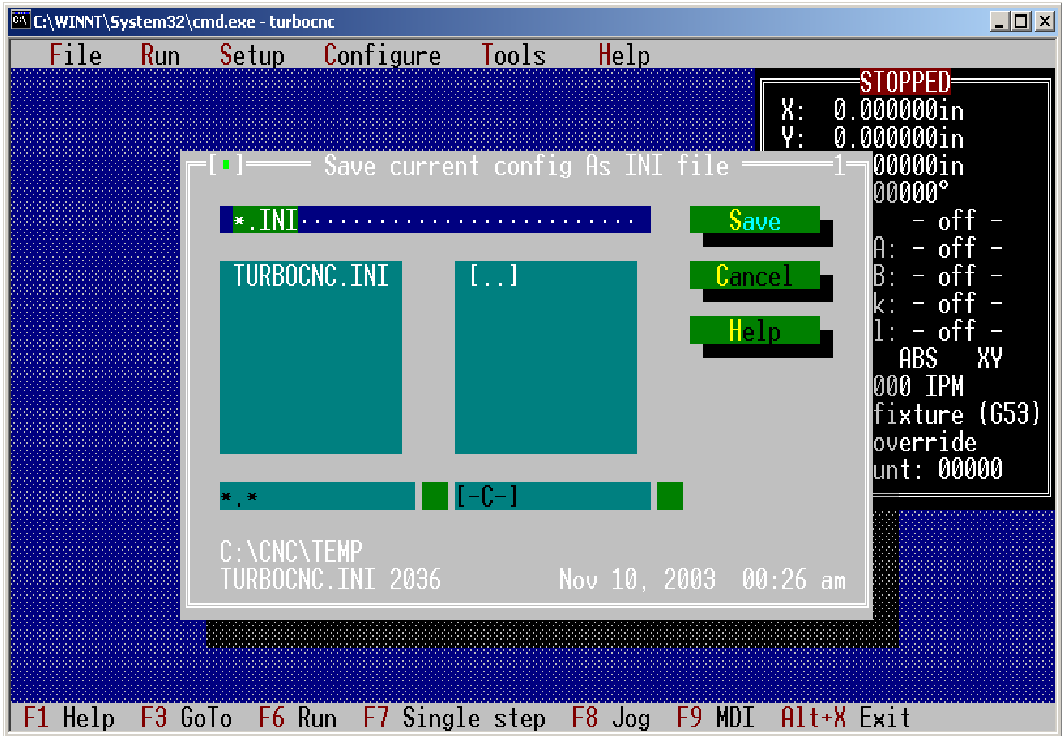
The arc IJK mode is the most common point of contention between TurboCNC and CAM programs.



Save Configuration…

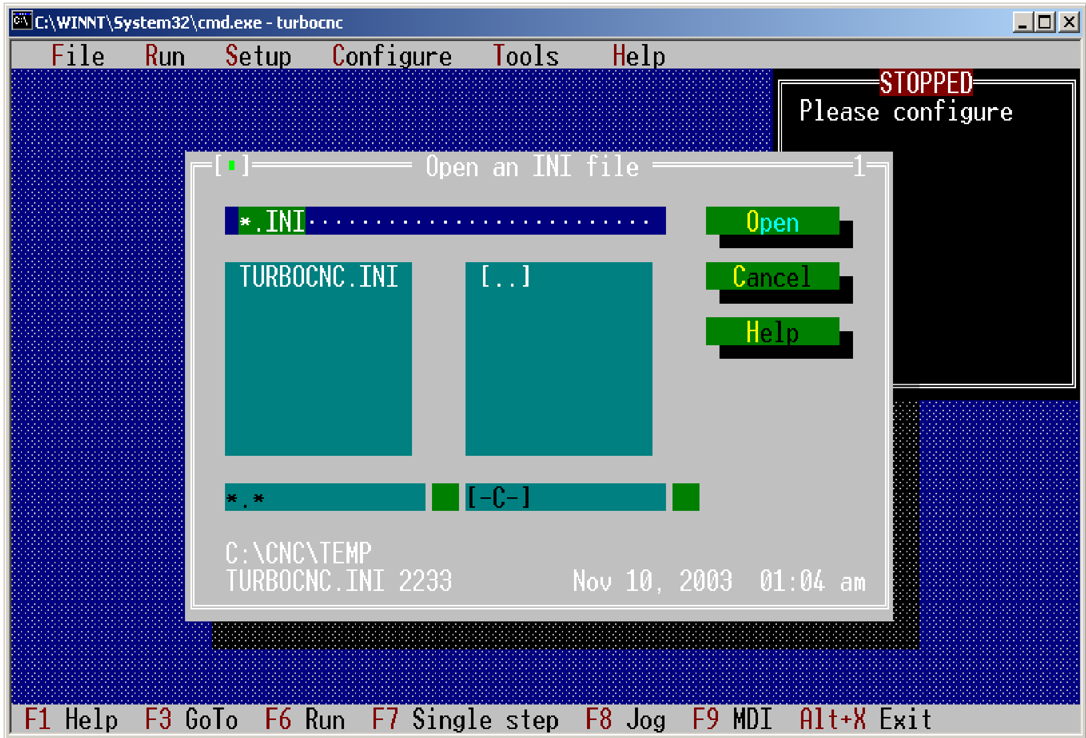
If a file with the same name as the currently loaded configuration file, with the extension '.old' exists, it will be deleted. The configuration file that was previously loaded will have its file name extension changed to '.old' and the current configuration will be written with the original name, in the original location. Upon completion a dialog box will appear to confirm that the file was successfully saved.

Save Configuration As…



Load New Configuration…

Upon selection a dialog box will open enabling you to select a configuration file to load. After the file has loaded a dialog box will appear, verifying that the file was loaded correctly and that the ports have been reset to use the loaded values. The status display is then updated to reflect the newly loaded values.



Reset Ports

'***Reset Ports***' reconfigures the software to use the currently configured parallel port I/O pin assignments. A dialog box will appear confirming that this has been done and offering the opportunity to save the current configuration. The same software routine as '***Save Configuration As***' is used.

Tools

Calculator

A simple calculator is provided to assist with setup calculations.

Help

There are a few options that provide you with access to critical documentation while you're still in the program. All the information is the same as what's in this manual - it's there for convenience on DOS machines that may not be able to read these files.

Introduction…

Provides an introduction to CNC machining and the fundamentals of the RS-274D language.

G-Code Ref…

A reference of the preparatory functions supported by TurboCNC.

M-Code Ref…

A reference of the miscellaneous functions supported by TurboCNC.

Programming…

Provides a reference on the use of the extended programming capabilities of TurboCNC, including variables, expressions (and the implemented mathematical functions), conditional programming (if…then) and operator interaction (ask/say).

What's New…

This section provides a quick overview of the new features of this version.

Shareware…

This item is a quick review of the principles behind 'Shareware', and instructions on how to register TurboCNC.

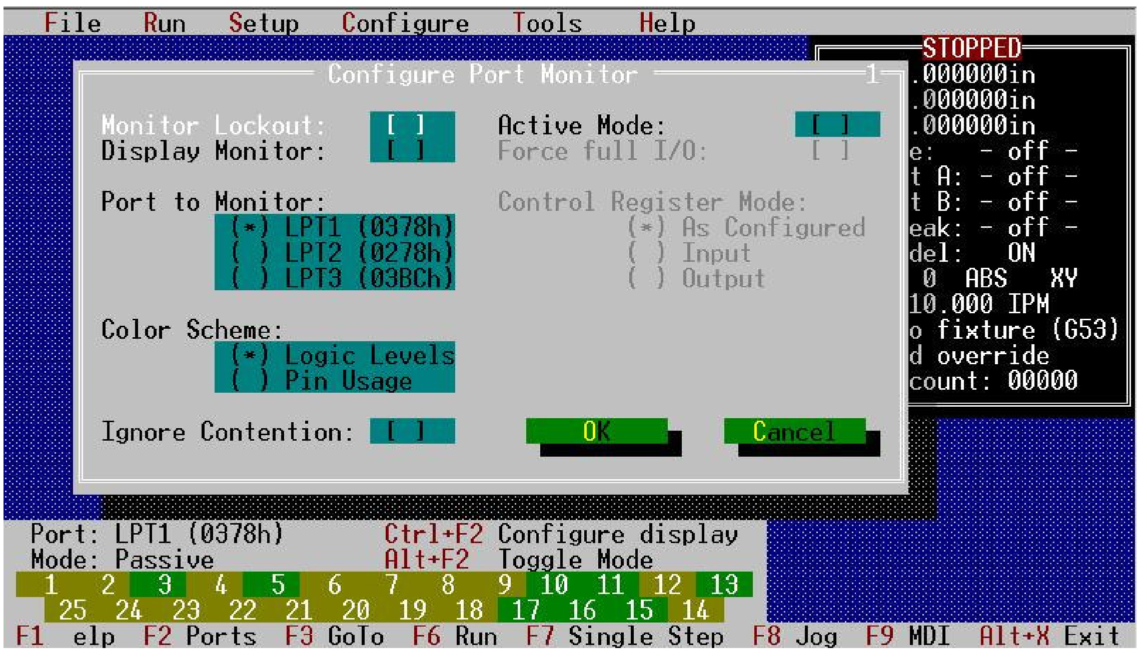
About…

Here's a list of the guilty parties who brought you this release of TurboCNC.

Port Monitor

The TurboCNC Port Monitor can be used to help set up a new CNC machine or troubleshoot problems in an existing installation. It is capable of displaying the state of the selected parallel port as known by TurboCNC in the Passive Mode, or altering the state of the output pins in the Active Mode.

The Port Monitor is displayed with F2, its configuration menu with Ctrl+F2, and its mode of operation is toggled by Alt+F2. None of the Port Monitor settings are saved in the configuration file. The default settings, and display are shown below.



Monitor Lockout:

Checking this option will lockout the Port Monitor, and clear ithe display. Use of the SAY command in a CNC program will automatically lockout the Port Monitor. Clearing the option will re-enable the Port Monitor function. It can then be re-displayed, either by checking the Display Monitor menu item, or with the F2 key.

Display Monitor:

This item is provided as a convenience allowing the Port Monitor to be displayed following a lockout from this menu.

Port to Monitor:

The monitored Port (LPT) can be selected using this menu item.

Color Scheme:

Two color schemes are provided.

Logic Levels

This scheme indicates the Logic Level through use of the background. Green is a high logic level, and brown is a low logic level. Pins that are contended (used for two or more

I/O devices) are shown with a red background. In this case the logic level is indicated by the foreground color as described in Pin Usage.

Pin Usage

The background color is used to indicate the configured use of the pin. Logic levels are indicated with the foreground color. A foreground color of green is a logic high, and white is a logic low. The appropriate I/O point must be enabled to show the pin usage. The following partial screen-shot shows the scheme:

Background color use is as follows:

- Red – Pin is contended (used for two or more I/O devices)
- Blue – Pin is configured for use as an output
- Cyan – Pin is configured for use as an input
- Brown – pin is not used

Ignore Contention:

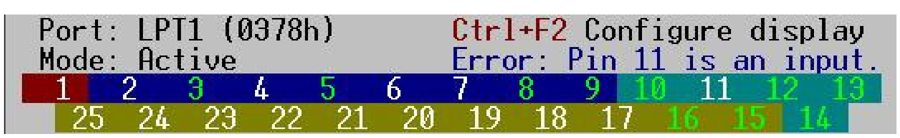
This option 'gets the red out' for those deliberately sharing an I/O pin between two or more devices. The scheme precedence for contended pin will is output (including step / dir, phase, and single point I/O functions) followed by input. If a pin is defined for use with both an output and input device it will be shown as an output.

Active Mode:

When checked the Port Monitor is placed in Active Mode. The mode can also be toggled using Alt + F2. Active Mode reads the ports, updating the display every half second, and allows the operator to modify the output pin state of the selected port as follows:

- keys 1 to 0 toggle pins 1 to 10 respectively
- keys Shift + 1 to Shift + 7 toggle pins 11 through 17
- left-clicking the pin number on the display will toggle that pin
- right-clicking an output pin number sends a slow (1 Hz) pulse train to that pin
- Ctrl + P will output a slow (1 Hz) pulse train on the last pin that was toggled

Note: Attempting to toggle an input pin results in an error message displayed as follows:



The message will automatically disappear when:

- 10 seconds has elapsed
- an output pin is toggled
- the Port Monitor mode is changed

An output pulse train of 1 Hz is toggled on or off, at the last pin that was toggled,.using Ctrl + P. The pulse train can also be sent to an output pin by right-clicking it. The pulse train is halted by toggling the function using Ctrl + P, or toggling another output pin with either the keyboard or the mouse. This feature can be used for cable identification, testing relays and indicators, or other fault-finding purposes. Pulse trains with higher frequencies can be generated in Passive Mode by defining an axis with its step pin set to the desired output, and using MDI to move the axis.

Active Mode is automatically changed to Passive Mode upon any of the following conditions:

- any attempt is made to execute RS-274 code (including internal codes executed by TurboCNC)
- the Port Monitor is locked out
- the Mode is changed
- the Port Monitor display is turned off

In ***Passive Mode*** TurboCNC normally only reads the ports as required to execute RS-274 code. Enabling the Port Monitor forces the ports to be read every time a block is executed. To force a port read and Monitor update without running a CNC program, bring up the MDI and hit 'Enter' each time you want to update the display.

When switching to Passive Mode, the ports are set back to their configured states and reset if required.

Force Full I/O:

Available only in Active Mode, this option forces TurboCNC to write to, and read all registers on all of the ports. During normal operation, and with the Port Monitor in Passive Mode, TurboCNC reads from and writes to only the ports and registers within the ports required by the current configuration. When switching to back Passive Mode, this option is deselected, and the ports reset if required.

Control Register Mode:

Available only in Active Mode, this option switches the mode of the control register pins (1. 14, 16,17) on all ports to the selected mode. Use caution when changing this option as improper use can damage your Printer Port or any connected devices. It is highly recommended that the CNC machine be physically disconnected from the port before changing this option. The background color for the pins will change to reflect their selected use if the Pin Usage color scheme is active. When switching to back Passive Mode, the pins routed through the control register are set back to their configured state, and the ports reset if required.

WARNING !	Disconnect your CNC machine before changing this option. Failure to do so can damage your Printer Port or any connected devices.

TurboCNC Configuration File

The TurboCNC configuration file, also know as the ini, provides a means of saving the program's state between runs. As its name implies all configuration information is stored in this file.

TurboCNC attempts to load a configuration file from the same directory as it is in, with the same name as the program. If the name 'turbocnc.exe' has not been changed, it will try to load 'turbocnc.ini', but if the name were changed for example to 'tcnc4.exe', it would attempt to load 'tcnc4.ini'.

If the configuration file is specified as the first parameter on the command line, TurboCNC will attempt to load the file specified. This is very useful if more than one CNC machine can be connected to the computer. Configuration files can be created for a lathe and a mill. The desired configuration can then be loaded by entering 'turbocnc lathe.ini' or 'turbocnc mill.ini'.

Although the configuration file can be edited using any text editor, including the file editor built into TurboCNC this is not recommended. The menu system should be used to modify the configuration file. If the currently loaded configuration file is edited, new configuration must be loaded using the menu item 'Configure / Load New Configuration', before exiting or the new values will be overwritten.

There are a few special parameters in the .ini file that are not accessible through the menu system, mainly obscure commands and setup options. These are:

UsePentiumTimer=NO Change this to YES to use the Pentium timer mode instructions. This can give you a significant performance increase (a factor of 4 in maximum pulse rate is typical) on a Pentium machine. By default, this is in compatibility mode. **Note:** The Pentium Timer currently cannot be used if TurboCNC has been compiled in the 'Protected Mode'. If set to 'YES' in the configuration this will be reset to 'NO' during protected mode startup for safety reasons.

[Jog_KeyBoard] Add this section to the cofiguration file if keys should be mapped to another keycode during jogging. The primary purpose of this capability is to allow compensation for foreign language keyboards. It can also be used to switch jog functionality to another key. The labels on the Jog menu are ***NOT*** changed through use of this capability.

MappedKey*xxx*=*kkk*:*ddd* is the format for entries under this section. The parameters are:

xxx – a number between 0 and 127, duplicates are not allowed. The keycode for the pressed key will be substituted when the file is re-written (saved).

kkk - The key that is pressed. This may be either the name from the table below, or a numeric keycode between 0 and 127.

ddd - The numeric keycode or name from the table below of the key that is to be substituted for the pressed key.

Note:

- Parameters kkk and ddd will be changed to the key's name from the table below if one is available when the configuration file is saved.
- Use the –Debug command line switch to detect invalid key names. The Keys being mapped will be reported in the debug file along with any mapping errors.

Name	Code	Name	Code	Name	Code	Name	Code
kyA	30	ky0	11	kyAlt	56	kyLeftArrow	75
kyB	48	ky1	2	kyAsterisk	55	kyLShift	42
kyC	46	ky2	3	kyBackSpc	14	kyMinus	12
kyD	32	ky3	4	kyBkSlash	43	kyNumLock	69

kyE	18	ky4	5	kyCalc5	76	kyPeriod	52
kyF	33	ky5	6	kyCapsLock	58	kyPgDn	81
kyG	34	ky6	7	kyColon	39	kyPgUp	73
kyH	35	ky7	8	kyComma	51	kyPlus	13
kyI	23	ky8	9	kyCtrl	29	kyQuote	40
kyJ	36	ky9	10	kyDel	83	kyRbracket	27
kyK	37			kyDownArrow	80	kyReturn	28
kyL	38			kyEnd	79	kyRightArrow	77
kyM	50			kyEsc	1	kyRshift	54
kyN	49			kyFwdSlash	53	kyScrollLock	70
kyO	24			kyGrayMinus	74	kySpacebar	57
kyP	25			kyGrayPlus	78	kyTab	15
kyQ	16	kyF1	59	kyHome	71	kyTilde	41
kyR	19	kyF2	60	kyIns	82	kyUpArrow	72
kyS	31	kyF3	61	kyLBracket	26		
kyT	20	kyF4	62				
kyU	22	kyF5	63				
kyV	47	kyF6	64				
kyW	17	kyF7	65				
kyX	45	kyF8	66				
kyY	21	kyF9	67				
kyZ	44	kyF10	68				

Command Line Options

The format of TurboCNC's command line is as follows:

```
turbocnc [inifile] [-tools toolfile] [-run partfile]
[-m] [-quick] [-nopos] [-debug]
```

No options are required on the command line in which case TurboCNC will attempt to load the default configuration (.ini) file, and the default tool file (if specified in the configuration file).

The options are:

inifile a valid configuration. TurboCNC will load this file and configure itself as specified. This is very handy for individuals using their computers to control several different systems (one at any one time) or with multiple configurations for the same system. The file must have the extension .ini.

-tools *toolfile* will attempt to load the specified tool file containing tool and fixture offsets.

-run *partfile* skips all menus and starts machining the part.

-m starts TurboCNC in the monochrome mode.

-quick skips the wait for keypress at the end of the startup screen, and proceeds directly to the main menu.

-nopos prevents TurboCNC from saving the positions and backlash sense of the axes when you exit the program, thus the old positions will be preserved. This is useful for those performing tests that run the parts file without a machine connected.

-debug writes debugging information and error messages to the debug.txt file located in same directory as TurboCNC.

NOTE: all files can be specified in file name (turbocnc.ini) or path/file (c:\cnc\turbocnc.ini) format. The filenames are limited to the DOS 8.3 format. See your operating system's instructions for the DOS 8.3 equivalent if you use programs that generate long file names to prepare your G-Code.

Part 3 – RS 274 Programming Guide

Introduction

An RS 274D program consists of lines of code. Each line is referred to as a Block. Blocks consist of a series of Words that define the operation to be performed. Each Word consists of an OpCode that specifies what the word refers to, and an Operand with the details. OpCodes are normally alphabetic, and operands are numeric. Beginning with release 4.0, TurboCNC allows variables or expressions to be used as the Operand of a Word.

Comments are used within RS 274 programs to make them easier for humans to read and maintain. Comments may be enclosed in brackets, or placed at the end of a line preceded by a semicolon. (Note: The TurboCNC parser will allow comments surrounded by brackets to be inserted between the OpCode and Operand of a Word. This practice is NOT recommended.)

Sample Program:

```
; This is a comment, preceded by a semicolon
(This is also a comment, surrounded by brackets)
M03 F5.0        ; This Block consists of two words
M05             ; OpCode = ‘M’, Operand = ‘05’
M03 F5.0        ; ‘F’ Operand specified as a value
M03 F#3         ; ‘F’ Operand specified as a variable
M03 F[6/2]      ; ‘F’ Operand specified as an expression
M02             ; End of Program
```

TurboCNC Parser

It is important to note that TurboCNC parses a line from left to right, resolving variable names and expressions as it encounters them. It ceases parsing the line upon the first error that it encounters. Parsed data is then stored in a line data structure.

Data is retrieved from the line data structure by the execution sequencer. The sequence is as follows:

- ‘M’ Words
- ‘T’ Words
- ‘G’ Words (includes ‘F’ words for G00-03, 28-32, 50, 76-78, 81-83, 178, and G183)
- ‘S’ Words
- ‘F’ Words

All words in each group are executed in the order received (left to right from the block) prior to moving to the next group.

General rules:

- More than one G or M word per block can be used, as long as there are no shared parameters.
- N words are ignored except when using jumps or subroutines.
- When using subroutines (see M98, M99), a unique N word is required on both the calling line and the return line.
- G, T, and F words are modal in general, with some exceptions.
- I, J, and K words for circular interpolation are incremental by default. This can be configured if your CAM program requires an alternate convention.

- Circular and helical interpolations are called using I, J, and K, or R notation, for any combination of axes. The plane selector (G17-19) should be called beforehand.

All moves involving more than one axis are interpolated, even in rapid. On some mill controls, the Z-axis will always retract first before moving X and Y. Not so for this program.

Angular axes are always driven in degrees modulo 360. For example, if a table is at 10deg and you command it to 350deg, it will go around "the long way". If you commanded it to -10deg instead, it will zip over to -10deg "the short way" and then report the current position as 350deg. Commanding a +720deg incremental move will index the table around two full revs, but the coordinate will remain unchanged since it is always set to somewhere between 0 and 360 only.

If you use IPR feeds through a G95 call, use the S word to set the spindle speed in RPM from which the feed rate will be calculated first. Alternately, call M50 to read the spindle speed if you have an encoder configured. The code to control the spindle directly from the computer is "empty" - this is so the users can add it themselves.

Metric distances, feeds, etc, are in mm and mm/min or mm/rev as appropriate.

Dwells are programmed in integer seconds by default, using the P word. This is configurable. Exception: The dwells for G82, G78, and G83 are in milliseconds.

The feed rate in an interpolated move is based on the actual distance covered by each of the linear axes involved. If there are none, then the first angular axis on the list is assigned the feed rate (degrees/sec).

If you hit ANY limit switch, the machine won't move again until you either disable the switch, or use Jog mode to back away. These are for preventing "going off the rails" only - use the home switches for calibrating the machine.

When block delete is active, a line with a "/" as the first character is skipped. Any word in a line that is preceded by a "/" is not executed. For example

```
G01 X1.234 F5 /F0.5
```

In block delete mode, the F5 is read as the feed rate and the second F word is ignored. In normal mode however, the F0.5 will override F5 on this line, and so the feed rate will be much slower.

OpCodes

OpCode is the term used when referring to a code that could refer to either a function or address.

Code	Meaning
G	Preparatory function
M	Miscellaneous function
N	Line (sequence) number
F	Feed rate
I	Interpolation parameter parallel to X axis, infeed parameter, 1st axis
J	Interpolation parameter parallel to X axis, 2nd axis
K	Interpolation parameter parallel to Z axis, thread lead
T	Tool offset select
R	Arc radius, release plane
S	Spindle speed
#	Assignment Operator / Numeric Variable Name
Q	Canned cycle peck increment
O	Line Number for M98 Subroutine Call and M97 Jump
P	Dwell duration

Operands

Dimension, sequence or other data following an OpCode is jointly referred to as Operand. The ability to use values, expressions or variables as operands greatly enhances the capabilities of TurboCNC.

Values

Simply put - values are numbers. While parsing a line, TurboCNC interprets any Operand that begins with a numeral from 0 to 9, a period, a plus sign '+', or a minus sign '-' as a value.

Expressions

An expression is a series of values and variables along with the operations and functions that must be performed on them to determine the operand. Expressions must be enclosed within square brackets, and may not be nested. (An expression may not contain sub-expressions; instead brackets are used within expressions to change the order of operations and indicate the parameters to be passed to a function.

Variables

Variables are values stored by TurboCNC, named with a pound sign '#' followed by an Integer from 1 through 9999. Variables with names of #1 through #999 are persistent, meaning that their values are stored in the configuration file upon exiting TurboCNC and re-loaded when TurboCNC is restarted. Variables with names from #1000 through #9999 are transient. These values are not restored when TurboCNC is restarted.

Variable names may be specified as values, variables, or expressions, and are resolved prior to retrieving the value. This allows advanced users to implement data structures such as arrays.

NOTE: Expressions and variables are not allowed as operands for the 'G', 'M', 'N', or 'T' OpCodes, instead these must be expressed as values.

Conditional Execution

The addition of the IF statement to TurboCNC, combined with variables and expressions has added the rudimentary programming capability of RS-274 D. Examples where this ability can be used to advantage are:

- cutting a contour in multiple passes – until you are through the material
- change feed rates when cutting a part out of different materials
- manufacturing the part held by only some of the fixtures of a complete setup

Preparatory Functions (G-Codes)

Supported Preparatory Functions

Code	Function	Code	Function
G00	Rapid positioning	G70	Inch units
G01	Linear interpolation	G71	Metric units
G02	CW circular interpolation (3D)	G72	CW helical interpolation (obsolete)
G03	CCW circular interpolation (3D)	G73	CCW helical interpolation (obsolete)
G04	Dwell	G76	Multi-pass threading cycle
G16	Set implicit planes (obsolete)	G77	Turning/Boring cycle
G17	Set XY plane	G78	Peck motion (general)
G18	Set XZ plane	G80	Cancel canned cycle
G19	Set YZ plane	G81	Drill cycle
G20	Inch units	G82	Drill cycle with dwell
G21	Metric units	G83	Drill cycle with peck
G28	Home all axes	G90	Absolute coordinates
G31	Probe move	G91	Incremental coordinates
G32	Probe cycle	G92	Preload of registers
G33	Single pass threading	G93	Inverse time feed rate
G50	Probe hole ID	G94	Inches/mm per minute
G53	Master coordinates (fixture 0)	G95	Inches/mm per rev
G54	Fixture offset 1	G97	Set spindle rpm
G55	Fixture offset 2	G178	Speed peck motion
G56	Fixture offset 3	G183	Speed peck drill cycle
G57	Fixture offset 4		
G58	Fixture offset 5		
G59	Fixture offset 6		

Table –1 Preparatory Functions, which are or have been supported by TurboCNC.

G00 Rapid Positioning

Function: Moves to a new position as fast as possible.

Syntax: G00 [axis words]

Example:

```
G00 X1.2 Y0.3  ; Moves to (1.2, 0.3)
```

Notes:

- Only the axes called out on the line will be moved.
- In a minor departure from the Standard, G00 is interpolated by default so that axes start and stop in unison and that a straight line is tracked between both positions. See the configuration section on how to change this.
- In Absolute Mode the coordinates given are absolute axis positions.
- In Incremental Mode the coordinates given are signed distances from the current positions.
- The absolute maximum speed is software limited to 2000 inches per minute (50 800 mm/min in metric) regardless of the physical limitations of the machine.
- The actual speed and ramping action of motion is chosen so as not to exceed the least capable moving axis.

G01 Linear Interpolation

Function: Moves to a new position linearly at some feed rate.

Syntax: G01 [axis words] [optional feed word]

Example:

```
G01 X1.2 Y0.3 F3.0  ; Moves to (1.2,0.3) at 3 units/minute
```

Notes:

The feed rate is calculated as follows by the program:

- A single linear axis feeds in units/minute or units/revolution depending on active modes. In G93 mode (inverse time feed rate) each move takes a constant amount of time to complete.
- A single angular axis feeds in degrees/second.
- Multiple linear axes feed according to the "true distance" of the move in units/rev or units/minute.
- Multiple angular axes feed in deg/sec for the first axis on the line, all others follow such that they start and stop in unison.
- Mixed angular and linear axes follow the rules for linear axes only, the angular axes will follow such that they start and stop in unison.
- No axis will ever go faster than the max speed established in setup.
- The Feed Rate Override modifies the feed rate, if enabled.
- Feed words are modal. If there is no feed word in the current block, the last feed word read will be used.

G02 CW circular interpolation (3D)

Function: Moves to a new position in a clockwise circular arc. The arc center is specified with signed offsets from the start position or implicitly by the magnitude of the radius.

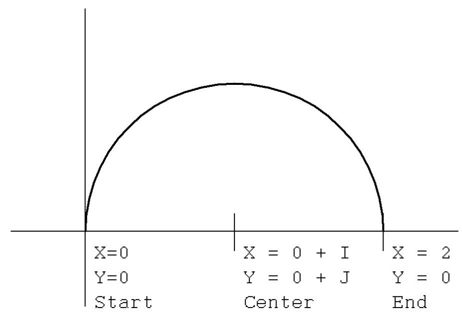
Syntax 1: G02 [two axis words (optional third)] [interpolation parameters] [optional feed word]

Syntax 2: G02 [two axis words (optional third)] [radius word] [optional feed word]

Example 1:

```
G17            (plane XY specified for clarity)
G00 X0 Y0      (get into start position)
G02 X2 Y0 I1 J0 F4
```

Moves from the current point to (0,0) in an arc with its center at the current point + 1 unit in the X direction at 4 units/minute. Absolute mode assumed. The "I" parameter is associated with the "X" axis, and the "J" parameter is associated with the "Y" axis.



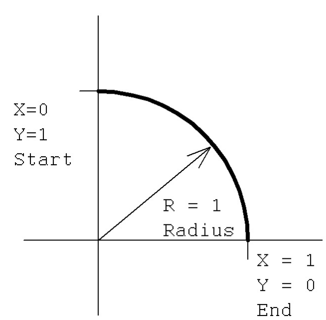

```
G17            (plane XY specified for clarity)
G00 X0 Y0 Z0   (get into start position)
G02 X2 Y0 Z1 I1 J0 F4
```

This generates the same arc as above, but adds linear movement along the Z-axis thus producing a helical cut.

Example 2:

```
G17          (plane XY specified for clarity)
G00 X0 Y1    (get into start position)
G02 X1 Y0 R1 F4
```

Moves from the current point to (2,0) in a clockwise arc of 1 unit radius less than 180 degrees at 4 units per minute. Again, absolute mode is assumed.




```
G17            (plane XY specified for clarity)
G00 X0 Y1 Z0   (get into start position)
G02 X1 Y0 Z1 R1 F4
```

This generates the same arc as above, but adds linear movement along the Z-axis thus producing a helical cut.

Notes:

I and J are incremental by default. (NOTE: In rev 3.00g and prior, the I and J word were absolute in absolute mode, incremental in incremental mode. This is no longer the case. See the dialect customizations section above for information on how to change this)

Using a negative radius chooses the arc greater than 180 degrees that crosses both points, a positive radius takes the arc that is 180 degrees or less. The program will stop with a warning if you have a radius that is impossible. The R form is notoriously inaccurate for arcs very close to 180 degrees.

The sense and 2D plane of the arc is set by the current plane mode. Axis words appropriate to the plane must be included i.e.:

- G17 – XY plane, X and Y [offsets I, J] [linear Z axis]
- G18 – ZX plane, X and Z [offsets I, K] [linear Y axis]
- G19 – ZY plane, Y and Z [offsets J, K] [linear X axis]

The order of parameters on the program line is not important. Use the I word for offset in the X direction, J for offset in Y, and K for offset in Z to describe the relationship of center position from the starting point. See the plane selector information section for more detail.

Start, destination and offset information for arcs can be specified using expressions, For example:

```
G02 X[3.5 + COS(135)] Y[4.0 + SIN(135)]
  I[3.5-(2.5 * COS(135))] J[4.0-(2.5 * SIN(135))]
```

This block assumes absolute mode for IJK, and has been split over two lines on this page. The arc lies on a circle with its center located at (3.5, 4.0), has a radius of 2.5 units, and the block will cause movement describing an arc clockwise from the current position to 135 degrees.

Full Circles:

If the destination letters are omitted, ***or,*** the distance between the start and end points of the arc is less than one full step for ***each*** axis on the plane, a full circle will be described by the tool motion. You must specify the move using the IJK form when omitting the axis end points for a full circle move, as the R form is indeterminate for these cases.

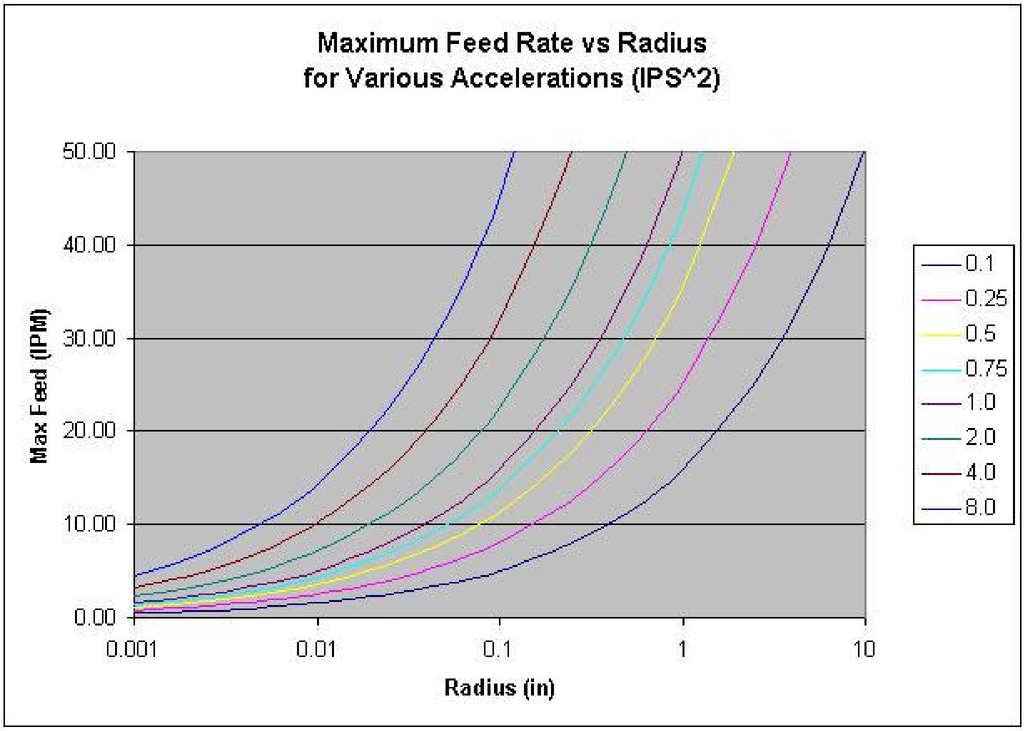
Feed rates in an arc:

Feed rates are applied along the helix (i.e.: cutter path) during a three dimensional move.

If the specified feed rate is higher than either the computer, or the CNC machine, is capable of achieving the feed rate will then be clamped at the system's maximum capability. The feed rate is clamped at the lower of that specified by Mav Vel, or that sustainable by the acceleration available at the axes. The formula used to determine the maximum feed rate based on acceleration is dependent upon the radius of the arc and is computed as follows:

```
f =  sqrt((Accel * Radius) / sqrt(2))
Accel is the product of Accel and Scale on the axis menu
```

The following chart shows the effect of radius on the maximum feed rate for various accelerations.



G03 CCW circular interpolation (3D)

Function: Moves to a new position in a counter-clockwise circular arc. The arc center is specified with signed offsets from the start position or implicitly by the magnitude of the radius.

See the section on G02 for general information on this code.

G04 Dwell

Function: Pauses execution for an integer number of seconds

Syntax: G04 [P word]

Example:

```
G04 P6 ; Six-second pause
```

Notes:

The P symbol and units of integer seconds are used. See the customization section for instructions on how to change the unit of time to milliseconds.

Any delay of more than 2 seconds will show a countdown.

Press a key to exit a delay prematurely.

WARNING	Never use G04 dwells and your hands as a tool-changer. It takes longer than you think... Use M00 or M06 instead.

G16 Set implicit planes

Obsolete Function: Set circular and helical interpolation plane to be defined implicitly by block

Background:

The original TurboCNC through rev 3.1a allowed an implicit plane specification in which the first two axes called out on a G02/03 line became the plane of action for the interpolation. In this mode, only the I and J parameters were used. The "I" offset was applied to the first axis on the line, the "J" axis to the second. This affected G02, G03, G72, and G73. For the helical moves, the third axis was always the direction of linear travel.

The idea here was to allow unusual interpolation schemes to be used on machines with many axes or non-conventional axis names.

This code is no longer supported! Use the conventional plane selectors (G17 through 19) for defining arc planes in new programs.

G17-19 Set current 2D plane

G17 Function: Sets plane for circular and helical interpolation to X-Y.

G18 Function: Sets plane for circular and helical interpolation to Z-X.

G19 Function: Sets plane for circular and helical interpolation to Z-Y.

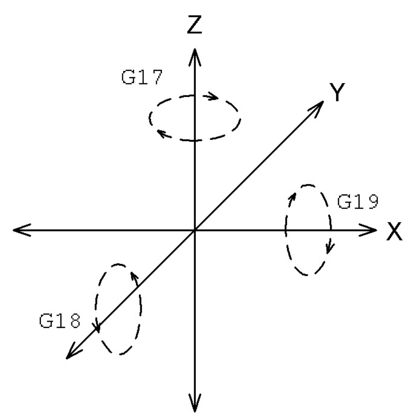
Syntax: none

Notes:

The rule for planes and how to determine the sense of CW vs. CCW is (from the Standard):

"Arcs shall be CW or CCW as viewed in the negative direction of the out-of- plane axis in a standard right hand axis system."

Translation: For mill users, if +X is rightward and +Y is toward the back of the machine (tool motion relative to the work!) and +Z is up, then as you look down (-Z) at the table in G17 mode, arcs will look CW or CCW as you'd expect.



For lathe users, if +Z is away from the headstock, and +X is increasing diameter, then in G18 mode CW/CCW will look reversed if you look down at the slide. This is because in this right hand system, -Y is looking up.

G20 Inch units

Function: Sets distance unit to inches.

Syntax: none

Notes:

All coordinates and feed rates will be in inches after this command is executed.

G20 and G21 are identical to G70 and G71.

Saving the .ini file while in either mode makes it modal on startup.

G21 Metric units

Function: Sets distance unit to mm.

Syntax: none

Notes:

All coordinates and feed rates will be in millimeters after this command is executed.

G20 and G21 are identical to G70 and G71.

Saving the .ini file while in either mode makes it modal on startup.

TurboCNC stores and uses all values in Imperial measure internally. This will cause very small conversion errors (> 0.00001 inch or 0.000004 mm) to show up on the display in Metric mode.

G28 Home all axes

Function: Drive each axis with a home switch to re-establish position.

Syntax: `G28 [optional feed word]`

Notes:

Each axis with a home switch will be driven to the home position where its coordinates will be reset. G28 is often used to recalibrate your machine during long cycle times where temperature creep or other position loss sources are a factor.

An axis must have a physical home switch to be "homed" in this context.

The mechanism is as follows:

- Each axis with a home switch moves simultaneously in the home direction.
- As each home switch goes active, each axis will stop
- When they are all on their switches, each axis will simultaneously move in the opposite direction
- As each home switch goes inactive, each axis will stop again
- Each axis position is re-established as configured in machine coordinates

The speed of the homing action is set by the modal F word and current feed mode.

It is not necessary to be in machine coordinates to initiate homing.

If no axis has a home switch, an error warning will be issued.

G31 Probe move

Function: Move to a position until the probe input goes active

Syntax: G31 [axis words] [optional feed word]

Example:

```
G31 Z-4.0 F10
```

Move to Z -4.0 at 10 IPM, until the probe input goes active.

Notes:

The move works in a similar fashion to G01. If the probe hits something, the machines stops and write the position to the file POINTS.DAT in the same directory as the executable.

If nothing is hit, the move ends at the position specified and no data is written.

Use G31 to get off of the part again if you need to, the program only accepts the inactive to active transition as valid input.

WARNING !	Note that if you move twice in the same direction with G31 or any other codes, you can smash your probe! Good practice is to design your probe with built in over travel that activates the panic stop input if exceeded.

G32 Probe cycle

Function: Repetitive probing to digitally scan a 2D or 3D surface

Syntax: G32 [axis bound 1] [bound 2] [optional bound 3] [discretization] [optional feed word]

Example:

```
G00 X0 Y0 Z0 ; get into position
G32 X1 Y1 Z-1 I0.250 F10
```

Moves X from 0 to 1 by .250 increments, each time moving Y from 0 to 1 by 0.250 increments. Z will plunge from 0 to -1 and back to 0 again, stopping and writing the position to disk for each probe strike on the downward stroke. Absolute mode assumed.

Notes:

- This is 2 or 3-axis general, so the last axis cycles the most and the first axis only once through a number of points determined by the discretization distance. You need at least two axes moving to scan. Using a single axis will move in the same way as G31.

- The digitizing scheme is of the "bed of nails" variety, e.g.: straight up and down in the last axis on the line.

- The output is written to a scan file in the same directory as the executable, with a filename of SURFSCAN.DAT.

WARNING !	Rounding error may prevent the last row or column from being scanned. Add a small amount to the bounding distance to avoid this:
	`G32 X1 Y1 Z-1 I0.250 F10 ;X1,Y1  ;may or may not be reached`
	`G32 X1.001 Y1.001 Z-1 I0.250 F10 ;This is better`

G33 Single pass threading

Function: Makes a single threading motion in sync with the spindle.

Syntax: G33 [axis coord] [lead parameter]

Example:

```
G33 Z-1.25 K0.050
```

Assuming inches and absolute mode, this threads 20 TPI to a shoulder at Z = -1.25

Notes:

A spindle index pulse must be enabled and present for this to work. See the section on hooking up a spindle encoder for details on how to do this.

The program will time-out after 5 seconds of spindle pulse inactivity. Similarly, the operation will be halted if the speed detected at the spindle is greater than 5000 rpm. This offers a measure of protection against the failure modes of signal noise, signal loss, or stalls.

K is the lead parameter for the Z-axis, I is for X, and J is for Y. See the dialect customization section if you want to specify pitches instead or change K for some other letter.

Multiaxis Operations: You can thread up to three axes simultaneously for tapered threading or unusual machining situations like gear hobbing. The lead and distance should work out to an equal number of revolutions for each. An axis with no lead specified will move at the regular feed rate.

Allow a few revs of "lead in" for the axis to synchronize. You need a lot of acceleration to keep up with the changes in velocity. If the axis falls behind and can't keep up, the program will stop and warn you.

The synchronization is always from the start point for each pass, so for multi-start threads, offset the start position by some fraction of the thread lead. For 30 degree infeed, change the start position on each pass along a 30 deg vector.

See also G76, multi-pass threading.

G50 Probe hole ID

Function: Finds the center of a hole using a touch probe

Syntax: G50 [optional feed word]

Notes:

Put the probe inside the hole approximately in the center, and call G50 from within a program or the MDI window.

This is an orthogonal six hit probing algorithm - clock positions struck are 12,6,9,3,12,6 in the XY plane as you normally view the part on a mill. You must have a touch probe installed and the probe input configured for this to work. Hole probing is always in the XY plane.

The speed of G50 probing is given by the modal feed word and mode.

Watch out for keyways and similar things that can throw you off. Use this command for auto-setting boring operations or fixture locating where some part-to-part variation is expected and you need to locate to a hole center exactly.

G53 Change to master coordinates

Function: Change to master coordinates, also known as fixture 0.

Syntax: none

Notes:

In master coordinate mode, no fixture offset is active. If tool 0 is set as well, you are in Machine Coordinates. These are the coordinates that are active while homing, and in setting up feed screw error compensation.

At startup, you are in machine coordinates by default. No offsets are active in machine coordinate mode.

G54-G59 Change fixture offset

Function: Change to a new fixture offset, from 1 (G54) to 6 (G59)

Syntax: none

Notes:

When the fixture offset is changed, all the tool offsets follow. Machining in a new offset mode is like having "zero" in a new place on your machine.

The coordinates you see on the screen are the master coordinates + the fixture offset in use + the tool offset in use. Tool 0 is no tool offset, G53 is no fixture offset.

The most common use for these is multiple fixtures on the same table. For example, home the machine in G53. Your usual 6" vise can be "zeroed" in G54 mode, and the center of a 5C collet vise, say, can be zero in G55. Changing modes puts you in a new coordinate system painlessly and allows different classes of parts to be machined on a large table.

G53-59 are modal. A fixture offset stays in effect until a new one is commanded.

G70 Inch mode

Function: Sets distance unit to inches.

Syntax: none

Notes:

Same as G20.

G71 Metric mode

Function: Sets distance unit to mm.

Syntax: none

Notes:

Same as G21.

G72 CW helical interpolation

Obsolete Function: Was similar to G02, but allowed a third axis to travel linearly as well.

Background:

Originally (through version 3.1a), TurboCNC used G02 and G03 as strictly 2D arc functions, and a separate G word for helical interpolation. This was unnecessary, and violated the Standard.

This function is no longer supported. Use G02 with a third axis callout instead.

G73 CCW helical interpolation

Obsolete Function: Was similar to G03, but allowed a third axis to travel linearly as well.

This function is no longer supported. Use G03 with a third axis callout instead.

G76 Multi-pass threading

Function: Machines an OD or ID thread completely on a lathe.

Syntax: G76 [X coord] [Z coord] [K height] [D first pass] [F lead] [A tool nose angle]

Example:

```
G76 X-0.210 Z-1.25 K.040 D0.003 F0.050 A60
```

Assuming inches and absolute radius mode, this threads 1/2-20 TPI UNF to Z = -1.25 completely.

Notes:

The starting point is returned to after each pass. The X location of the starting point determines the back out distance after each cut it complete.

The A word specifies the included tool nose angle, which is customarily 60 degrees for common threads. The tool will automatically feed on an angle of 1/2 A. If A is not specified, it defaults to 0, which produces a radial infeed.

K, D and F are always positive, regardless of the actual orientation or "hand" of the thread.

Each successive pass will remove the same area of material as the first pass did, in order to equalize the torque load. This is standard, and helps to improve the finish.

G77 Turning/Boring/Milling Cycle

Function: Turning with multiple passes, infeeding by an amount set by the I word.

Syntax: G77 [repetitive axis] [infeed axis] [feed per pass] [optional feed rate]

Example:

```
G77 Z-1.250 X0.250 I0.050 F5.0
```

On a radius programmed lathe that cuts to an X coordinate of 0.250, back and forth in Z from current position to -1.250, feeding in 0.050 each pass and cutting at 5 units/min. Here's the actual motion sequence for on the above, assuming the tool started at 0,0:

X0.050 slowly
Z-1.250 slowly
X-0.025 slowly
Z0 rapidly
X0.100 slowly
Z-1.250 slowly
etc...

Notes:

The order of the operands defines the motion. The back-and-forth action will occur from the current position of the first axis on the line to the position specified. The second axis called out will eventually reach the position specified by feeding in increments of I each pass.

The sign of I is unimportant. Always use I for the infeed, no matter what other axes are being used.

Infeed is applied to the second axis on the line. Creative use of this command can be made for fly cutting / slab-milling or cutting deep blind slots on mills or trimming the edges of stock.

After each pass, the control "backs off" by 1 1/2 times the infeed. So if you're boring, make sure you have enough clearance for the bar.

If the infeed doesn't divide into an even number of passes, a small finishing pass will be taken.

G78 Peck Motion Cycle

Function: Feeds to a position on an axis, incrementally in "pecks" with a full rapid retract.

Syntax: G78 [axis of action] [peck distance] [optional feed word] [optional dwell]

Example:

```
G78 Z-2.000 I0.100 F2.0
```

Moves from the current position to Z=-2.000 at 2 units/min 0.100 units at a time (retracts to original Z each time).

Notes:

The sign of I is unimportant, but this code is always used with I, regardless of the actual letter for the moving axis.

If you put in a dwell parameter with the P operator, a dwell in milliseconds will be taken at the end of each pass.

```
G78 Z-2.000 I-0.100 F2.0 P100
```

does the same as the previous example, but with a tenth of a second dwell at the bottom of the hole. This helps the tool life considerably in some cases. After each peck and retract, the tool will rapid to 10% of the peck increment before the last stop, and then feed in again. The letter and units used for the dwell are the same as for the G8x cycles if user configured.

This command is single axis only.

A good use of this code is "peck turning" plastic on a lathe to keep the chips short.

G80 Cancel drill cycle

Function: Cancel canned drill cycle

Syntax: none

Notes:

It's good practice, but not strictly necessary, to put this code after a series of the G81, G82, or G83 drill cycles. It clears the canned cycle variables from memory.

Some CAM programs automatically generate this code after every series of drilled holes.

G81 Drill cycle

Function: Drill a hole on a 3-axis mill

Syntax: G81 [axis words] [release plane] [optional feed word]

Example:

```
G81 X1 Y1 Z-0.75 F2.0 R0.25
```

The following will occur. Absolute mode and inch units are assumed.

Rapid to R plane if Z is less than 0.25 absolute

Move the table to the XY position (1,1) specified; holding Z at the point it was before.

Feed the Z-axis to -0.75 at 10ipm.

Rapid the Z-axis to 0.25 (the release plane)

Notes:

This is the canonical RS-274D drill cycle. Drills a hole at a specific XY position, to a depth Z at the current feed rate, and retracts to a release plane "R".

To drill another hole just like the first, just input the XY position on the next line:

```
G81 X1.5 Y1.25
```

This second hole will be done the exact same way was the first, but at the new position of X=1.5 and Y=1.25.

Release plane: If the R plane is between the current Z position and the bottom of the hole, the control will rapid to the R plane after moving XY and before drilling. If the R plane is "above" where the drill is in Z at the start of G81, the control will rapid to the R plane first before moving XY. This affords maximum safety without overly compromising speed.

Note that R is absolute in absolute mode, incremental in incremental mode! All the other parameters behave similarly.

All of the coordinates (XYZR) need to be called out on the first G81. These will "stay in effect" thereafter until G80 is called (modality). So, if you have a series of holes that are all to the same depth, you can use this:

```
G81 X1 Y1 Z-0.75 F2.0 R0.25 ;First hole
G81 X2 Y2
G81 X3 Y2
X2.5
G80 ;Four holes were drilled at (1,1) (2,2) (3,2) and (2.5,2)
```

These cycles ignore the plane selector as per the Standard. The drilling always occurs in Z, the positioning in XY.

G82 Drill + Dwell cycle

Function: Drill cycle with dwell at the bottom of the hole

Syntax: Similar to G81, but requires a # parameter for the dwell at the bottom of the hole in milliseconds.

Example:

```
G82 X0 Y0.5 Z-1 F10 R0.25 P250
```

Drills at coord (0,0.5) to a depth of Z=-1 at 10 IPM. Dwells for a quarter second, and then retracts to Z=0.25.

Notes:

Dwell is in milliseconds for this code by default. This can be customized to seconds, or some other letter chosen besides P for CAM compatibility.

G83 Peck drill cycle

Function: Drill with a pecking action and an optional dwell

Syntax: Similar to G81/82, but requires an I parameter for the peck increment when drilling.

Example:

```
G83 X0 Y0.5 Z-1 F10 R0.25 Q0.100 P250
```

This is the same as the G82 example above, but the drill will descend at the 10 IPM feed rate in 0.100 inch pecks with a rapid retract to the original Z. The drill rapids down to 10% of the peck distance above the bottom of the hole before feeding again to minimize "air time".

Notes:

The dwell parameter (P) is optional with G83 as a convenient departure from the Standard. The above example will dwell a quarter second at the end of each peck. The dwell is in integer milliseconds by default.

The sign of Q is unimportant. The letters P and Q, and the dwell units are customizable.

G90 Absolute coordinates

Function: Set coordinates to absolute mode (default).

Syntax: none

Notes:

In absolute mode, all axis words and many parameters refer to an absolute coordinate position.

G90/91 are modal. Either mode will stay in effect until the other is called.

G90 is the default mode at startup.

G91 Incremental coordinates

Function: Set coordinates to incremental mode (offsets from current position)

Syntax: none

Notes:

In incremental mode, all axis words and many parameters are a signed offset from the current position.

G90/91 are modal. Either mode will stay in effect until the other is called.

G92 Preload of registers/Set machine coordinates

Function: Set position without motion

Syntax: G92 [axis words]

Example:

```
G92 X0       ;Zeroes X axis
G92 X0 Y0 Z0 ;Zeroes the principle axes on a mill
G92 Z1.234   ;Z is now set to 1.234
```

Notes:

This code sets the position of any or all axes to a specific value. Use this to reset the position inside a program. No motion will occur.

In machine coordinate (G53 T0) mode, the machine coordinates themselves are updated.

In any fixture offset mode (G54-G59) and T0, the fixture offset is updated.

In any tool-offset mode (T1-T20), the tool offset is updated.

This code is not modal.

G93 Inverse time feed rate

Function: Set feed rate unit to inverse time ratio

Syntax: none

Example:

```
G93 F60  ;All G01 blocks will take one second
G93 F120 ;All G01 blocks will take 1/2 second
G93 F0.5 ;All G01 blocks will take 2 minutes
```

Notes:

In this mode, the *length of time* that each block will take to execute is controlled. The unit is the reciprocal of length of time in minutes. (e.g.: 60 is 1/60th of a minute or one second).

G93/94/95 are modal to one another. Each mode will stay in effect until another is called.

This mode is useful for unusual situations where the feed rate is difficult to calculate directly, but the overall time to move is known - such as when several axis types are moving simultaneously.

As with the other feed modes, if the commanded feed rate is too fast for the machine, the motion will be performed at the fastest available speed.

With arcs, each arc segment is treated as an individual block in this mode.

G94 IPM feed rate

Function: Set feed rate unit to units/min

Syntax: none

Notes:

G93/94/95 are modal to one another. Each mode will stay in effect until another is called.

G94 is the default mode at startup.

G95 IPR feed rate

Function: Set feed rate unit to units/rev.

Syntax: G95 [optional S word]

Example:

```
G95 S1000 F0.002 ; Feed is now 0.002/rev at 1000 rpm
```

Notes:

Be sure to have set the "S" word for spindle speed when calling this function for the first time before any motion takes place. You can also read the spindle speed after engaging G95 mode by using the M50 code if you have a spindle encoder on your machine.

G93/94/95 are modal to one another. Each mode will stay in effect until another is called.

G97 Program spindle RPM

Function: Set spindle RPM

Syntax: G97 [S word]

Example:

```
G97 S1000 ;1000 rpm
```

Notes:

This function is a "placeholder" in the source code. Registered users can use it as a starting point to program their own spindle control routines into TurboCNC.

If you use units/rev feed rates by calling G95, you must use this code or otherwise set the S parameter with the spindle speed before trying to move.

G178 Speed peck motion

Function: Speedy generalized single axis pecking motion

Syntax: G178 [axis of action] [peck distance] [optional feed word] [optional dwell]

Example:

```
G178 Z-2.000 I0.100 F2.0
```

Moves from the current position to Z=-2.000 at 2 units/min 0.100 units at a time with dwells.

Notes:

This command is single axis only, and is identical to G78, except that no retracts are made while machining. Only the dwells are taken.

In some cutting situations you can use this to avoid dangerous continuous chips while machining, and saving a significant amount of time in the process by not retracting fully.

G183 Speed peck drill cycle

Function: Speedy peck drilling in pecks with dwells

Syntax: Same as G83

Notes:

This code operates in exactly the same manner as G83, except that it does not retract at all between each peck. The dwells are taken.

In some cutting situations you can use this to avoid dangerous continuous chips while conserving machining time if many holes are to be drilled.

Miscellaneous Functions (M-Codes)

Supported Miscellaneous Functions

Miscellaneous Functions are generally simpler than Preparatory Functions and seldom have parameters.

Code	Function	Code	Function
M00	Automatic halt	M40	Select Gear Ratio 1
M01	Optional halt	M41	Select Gear Ratio 2
M02	End of program	M42	Select Gear Ratio 3
M03	Spindle on CW	M43	Select Gear Ratio 4
M04	Spindle on CCW	M44	Select Gear Ratio 5
M05	Spindle off	M45	Select Gear Ratio 6
M06	Tool change	M46	Select Gear Ratio 7
M07	Coolant A on	M48	Restore feed override
M08	Coolant B on	M49	Cancel feed override
M09	Coolants off	M50	Read spindle speed
M10	Clamp	M60	Jump to Subroutine (obsolete – use M98)
M11	Unclamp	M62	Return from Subroutine (obsolete – use M99)
M13	Spindle CW and coolant A on	M70	Set PLC handshake output to inactive
M14	Spindle CCW and coolant A on	M71	Set PLC handshake output to active
M17	Enable drives	M72	Wait for PLC handshake input to go inactive
M18	Disable drives	M73	Wait for PLC handshake input to go active
M21	Open collet	M97	Jump
M22	Close collet	M98	Jump to subroutine
M30	End of tape	M99	Return from subroutine

Table –2 Miscellaneous Functions that are or have been supported by TurboCNC.

M00 Automatic halt

Function: Halts program until operator presses a key.

Notes:

Comments included in the block containing M00 are displayed within the prompt that is raised. A vertical bar, '|' can be used to force a new line. This feature is a convenient method of presenting instructions to the machinist at runtime.

M01 Optional halt

Function: Halts the program only if optional halts are enabled.

Notes:

Similar to M00. Use Optional Halts under the machining options menu to toggle whether these are enabled.

This is commonly used for "first article" type checks while the part is on machine.

Comments included in the block containing M01 are displayed within the prompt that is raised. A vertical bar, '|' can be used to force a new line. This feature is a convenient method of presenting instructions to the machinist at runtime.

M02 End of program

Function: Stops program execution.

Notes:

This (or M30) must be the last line of a program.

Comments included in the block containing M02 are displayed within the prompt that is raised. A vertical bar, '|' can be used to force a new line. This feature is a convenient method of presenting instructions to the machinist at runtime.

M03 Spindle on CW

Function: Turns on spindle in the clockwise direction.

Notes:

Clockwise is the usual direction for a lathe or mill. If the spindle is commanded to run forward while it is already on, it will be shut off for 7 seconds and then re-activated in the new direction.

M04 Spindle on CCW

Function: Turns on spindle in the counter-clockwise direction.

Notes:

Similar to M03.

Turn the spindle off with M05 before changing directions or calling M03/04. If the spindle is commanded to reverse while it is already on, it will be shut off for 7 seconds and then re-activated in the new direction.

M05 Spindle off

Function: Turns off spindle

Notes:

Turns off the spindle output line. No braking is assumed. The spindle direction line state is not affected.

M06 Tool change

Function: Halt for tool change

Example:

```
M06 T1 ; Stops with a prompts for change to tool 1
       ; and changes to the tool 1 coordinate offset
```

Notes:

This is essentially the same as M00, but with a prompt to tell the user which tool is being requested. Registered users may program their own, more sophisticated routines for automated tool changes here.

The T word is required with M06. It may also be placed on any program line to change the tool offset without pausing the program.

If the tool turret index line is enabled, M06 will not prompt the user, but activate the turret index line briefly instead. The default time period is two seconds, the maximum period is 120 seconds.

The delay period for the turret index line is set in the ini file or the configuration menu under the General heading.

Comments included in the block containing M06 are displayed within the prompt that is raised. A vertical bar, '|' can be used to force a new line. This feature is a convenient method of presenting instructions to the machinist at runtime.

M07 Coolant A on (flood)

Function: Turns on relay A

Notes:

This code is traditionally associated with flood coolant. It can be used as a generic output to control anything.

M08 Coolant B on (mist)

Function: Turns on relay B.

Notes:

This code is traditionally associated with mist coolant. It can be used as a generic output to control anything.

M09 Coolants off

Function: Switches off both relay A & B.

Notes:

Both coolant outputs (A and B, see M07 and M08) will be set to inactive.

M10 Clamp

Function: Closes a machine clamp

Example:

M10 Q12; Closes clamp twelve

Notes:

Use numbers 0-15 for the clamps. If Q is not specified, it defaults to clamp 0.

At a minimum, the clamp drive, direction, and closure sense lines need to be configured on the machine to use this command.

The clamp address appears on the four clamp select output lines in binary, if they are configured on the machine. Clamp 0 is all lines inactive; clamp 15 sets all lines active.

You do not need to configure all 4 clamp select lines. For example, if you only have four clamps to control then it is permissible to use the first two clamp select lines only.

The clamp will be driven until the closure line goes active, or when 15 seconds has elapsed, whichever comes first.

M11 Unclamp

Function: Identical to M10, except that it opens a machine clamp.

Notes:

At a minimum, the clamp drive, direction, and open sense lines need to be configured on the machine to use this command.

The clamp will be driven until the open line goes active, or when 15 seconds has elapsed, whichever comes first.

M13 Spindle CW and coolant A on

Function: Turns the spindle on in the clockwise direction and activates the "A" coolant.

M14 Spindle CCW and coolant A on

Function: Similar to M13, but the spindle will be started counterclockwise.

M17 Enable drives

Function: Set drive enable lines active

Notes:

The Stepper World SP3 and some versions of the MAXNC drives require an enable signal in order to operate. After the drive enable output lines have been configured in TurboCNC, use this code to turn them on.

The drives will be disabled automatically when:

- a panic abort occurs during motion
- you exit TurboCNC normally

They will be enabled when:

- a program is started or resumed
- you enter jog mode
- TurboCNC is started
- you input an MDI command

You can also control these manually by using the options under the setup menu.

M18 Disable drives

Function: Set drive enable lines inactive

Notes:

Similar to M17, this turns the drive enable lines off.

M21 Open collet

Function: Opens a collet

Notes:

When this code is executed, the Collet Open line goes active for a short time, and then returns to the inactive state. The default length of time is two seconds and the maximum is 120. Change the length of time allowed by editing the ColletOpenTime(ms) entry in the ini file.

No feedback is required from the actual machine. This is designed to work with a pneumatic collet closer on a lathe.

The delay periods for M21 and M22 are set in the ini file or the configuration menu under the General heading.

M22 Close collet

Function: Closes a collet

Notes:

Similar to M21, this code activates the Collet Close output line for some length of time. Two seconds is the default, 120 seconds is the maximum. Change the length of time allowed by editing the ColletCloseTime(ms) entry in the ini file.

M30 End of program & rewind

Function: Stops program execution

Notes:

Functionally identical to M02. Hard disks don't need to be "rewound" of course, but paper tapes used to be! Some CAM programs generate this code instead of M02 at the end of a program, so it's here for compatibility.

Comments included in the block containing M30 are displayed within the prompt that is raised. A vertical bar, '|' can be used to force a new line. This feature is a convenient method of presenting instructions to the machinist at runtime.

M40 – M46: Gear Changes

Function: Selects the speed map associated with the specified gear.

Notes:

Spindle speed control is oriented to the Sherline spindle. As such the default compilation only implements M40 through M43. Increase MaxRatios in the spdmap unit to 7 to take advantage of the full range of codes.

The following mappings are defined:

- M40 – Standard Pulleys, Low Range (45 – 1400 RPM)
- M41 – Standard Pulleys, High Range (70 – 2800 RPM)
- M42 – 10K RPM Pulleys, Low Range (150 – 2200 RPM)
- M43 – 10K RPM Pulleys, High Range (1500 – 10200 RPM)

M48 Restore feed override

Function: Restore feed override

Notes:

This brings the feed override back to whatever it was just before the most recent M49 call.

M49 Cancel feed override

Function: Cancel feed override

Notes:

This resets the feed override to 100% "from the inside". Use it before entering a critical section of your program that requires an exact feed rate. The feed override can be restored with the M48 code.

M50 Read spindle speed

Function: Set S word with actual spindle speed

Notes:

This reads the spindle speed into the program for use with per revolution feed rates (see G95) by reading the spindle index pulse.

You must have an index pulse enabled to use this code. See the hardware section for details on how to set this up.

The operation will time out if five seconds goes by without a signal from the spindle.

An error will be triggered if the spindle appears to be going faster than 5000 rpm.

M60 Jump to subroutine (*obsolete function)*

Function: Jump to a new block saving return

Syntax: [N word] M60 [jump target]

Example:

```
N0010 M60 O0100  ; Jump to subroutine 0100
M05              ; Subroutine returns here - spindle off
M02              ; End Program

N0100 M03        ; Subroutine to turn spindle on
M62              ; Return from subroutine
```

Notes:

This function is considered obsolete. Use of M98 is strongly suggested. It is included as a partial bridge to previous versions of TurboCNC.

The jump target word is set to 'O' by default. This can be reconfigured using the 'Configure/Dialect' menu.

Jumps to the line with the N word identical to the jump target word (N0100 in the example).

The N word of the line with the M98 is saved for the return (10 in the example). Naturally, this means you should have a different N word on both the calling line and the target line.

Subroutines can be nested 20 levels deep.

M62 Return from subroutine (*obsolete function)*

Function: Return to block immediately after the most recent M60 call

Syntax: M62

Example:

See M60

Notes:

This function is considered obsolete. Use of M99 is strongly suggested. It is included as a partial bridge to previous versions of TurboCNC.

The next block to be executed will be found on the line following the most recent M60 call.

See SAMPLE.CNC for a simple nested subroutine example. Please note that when using subroutines, you'll need a unique N code on each calling line so that the program knows where to go back to.

To visualize program execution with subroutines, imagine all the code in the subroutine invisibly pasted in beneath the calling line. None of the modal codes are affected.

M70 Set PLC handshake output to inactive

Function: Set PLC handshake output to inactive

Notes:

Two PLC handshaking lines can be configured in TurboCNC. These are for telling external logic, such as an automatic tool changer or what-have-you to do some work. M70 sets the output handshake line to the inactive state. This can also be used to control extra solenoids or relays.

M71 Set PLC handshake output to active

Function: Set PLC handshake output to active

Notes:

Similar to M70, M71 sets the output handshake line to the active state.

M72 Wait for PLC handshake input to go inactive

Function: Wait for PLC handshake input to go inactive

Notes:

When this code is called, the program stops running and waits for the PLC input line (separate from the output line) to go to the inactive state. This is intended to be used to synchronize the program with external logic that might be loading more stock, or performing some other function.

M73 Wait for PLC handshake input to go active

Function: Wait for PLC handshake input to go active

Notes:

Similar to M72, this pauses until the PLC line goes to the active state. With either of these two codes, the user can press a key to bypass the wait condition.

M97 Jump

Function: Jump to a new block

Syntax: M97 [jump target]

Example:

```
M97 O0200    ; Jump to block 0200
G00 X1       ; This line is skipped
N0200 M02    ; Jump target - Program End
```

Notes:

The jump target word is set to 'O' by default. This can be reconfigured using the 'Configure/Dialect' menu.

Jumps to the line with the N word identical to the jump target word (N0200 in the example).

Code following an M97 in a block is ***not executed.*** M-Codes within a block are executed first, in the order that they are encountered. Only the parameters that follow the M97 for M-Codes encountered earlier in the block will be used. Every other code or parameter is ignored.

The jump function is very useful when combined with 'Conditional Programming'

If there is a parsing error in the line containing the jump target, the message 'Jump Target not Found' will be displayed.

M98 Jump to subroutine

Function: Jump to a new block saving return

Syntax: [N word] M98 [jump target]

Example:

```
N0010 M98 O0100  ; Jump to subroutine 0100
M05              ; Subroutine returns here - spindle off
M02              ; End Program

N0100 M03        ; Subroutine to turn spindle on
M99              ; Return from subroutine
```

Notes:

The jump target word is set to 'O' by default. This can be reconfigured using the 'Configure/Dialect' menu.

Jumps to the line with the N word identical to the jump target word (N0100 in the example).

The N word of the line with the M98 is saved for the return (10 in the example). Naturally, this means you should have a different N word on both the calling line and the target line.

Subroutines can be nested 20 levels deep.

Code following an M98 in a block is ***not executed.*** M-Codes within a block are executed first, in the order that they are encountered. Only the parameters that follow the M98 for M-Codes encountered earlier in the block will be used. Every other code or parameter is ignored.

If there is a parsing error in the line containing the jump target, the message 'Jump Target not Found' will be displayed.

M99 Return from subroutine

Function: Return to block immediately after the most recent M98 call

Syntax: M99

Example:

See M98

Notes:

The next block to be executed will be found on the line following the most recent M98 call.

Code following an M99 in a block is ***not executed.*** M-Codes within a block are executed first, in the order that they are encountered. Only the parameters that follow the M99 for M-Codes encountered earlier in the block will be used. Every other code or parameter is ignored.

See SAMPLE.CNC for a simple nested subroutine example. Please note that when using subroutines, you'll need a unique N code on each calling line so that the program knows where to go back to.

To visualize program execution with subroutines, imagine all the code in the subroutine invisibly pasted in beneath the calling line. None of the modal codes are affected.

If there is a parsing error in the line containing the jump target, the message 'Jump Target not Found' will be displayed.

S-Word Handing:

The interpretation of the S-Word has been modified to control the DigiSpeed output. A count of the difference between the currently set count at the DigiSpeed, and that corresponding to the specified speed is sent to the DigiSpeed changing its output.

Setting a speed of zero will disable the Digispeed. This must be re-enabled with a M03 or M04.

Speeds below the minimum and above the maximum mapped values will result in an error report.

Programming Extensions

Expressions

An expression is a mix of values, variables, operators and functions, enclosed in square brackets, which is resolved to a single value that is substituted for the expression BEFORE the OpCode is executed. The normal rules of mathematics are used in resolving expressions. These are:

- contents of brackets are resolved from the inner most to the outer most level
- functions are resolved to a value
- powers are resolved in the order encountered from left to right
- multiplication and division are resolved in the order encountered from left to right
- addition and subtraction are resolved in the order encountered from left to right

Expressions are resolved as 'real' values accurate to 15 decimal places. The result is converted to a 'string' value and handled as if it had been encountered on the input line instead of the expression. Expressions may be used in the placed of values with the exception noted below.

NOTES:

- Expressions are not allowed as operands for 'G', 'M', 'N', or 'T' OpCodes.
- M97, M98, M99 will fail with a 'Target not Found' if there is an error in the expression

Operators

Operators perform a mathematical operation using two values. The operators available within TurboCNC expressions are listed in the following table:

Operator	Example	Explanation
+	a + b	b is added to a
-	a - b	b is subtracted from a
*	a * b	a is multiplied by b
/	a / b	a is divided by b
^	a ^ b	a is raised to the power of b
E	aEb	a is multiplied by 10^b (Scientific Notation)

Functions

Functions return a value based on their name or a single input value. The functions available within TurboCNC are:

Function	Example	Explanation
PI	PI	returns the value of PI correct to 15 decimal places
ABS	ABS(a)	returns the absolute value of 'a'
INT	INT(a)	returns 'a' as an integer rounded to the nearest whole number
SQR	SQR(a)	returns 'a' * 'a'
SQRT	SQRT	returns the square root of 'a' (the number which multiplied by itself yields 'a')
LN	LN(a)	returns the natural log of 'a'
LOG2	LOG2(a)	returns the log of a in base '2'
LOG10	LOG10(a)	returns the log of 'a' in base 10
EXP	EXP(a)	returns e raised to the power of 'a' (inverse of LN)
SIN	SIN(a)	returns the sine of the angle 'a', measured in degrees
COS	COS(a)	returns the cosine of the angle 'a', measured in degrees
TAN	TAN(a)	returns the tangent of the angle 'a', measured in degrees
COTAN	COTAN(a)	returns the cotangent of the angle 'a' measured in degrees
ARCSIN	ARCSIN(a)	returns the angle who's sine is 'a', in degrees
ARCCOS	ARCCOS(a)	returns the angle who's cosine is 'a', in degrees
ARCTAN	ARCTAN(a)	returns the angle who's tangent is 'a', in degrees

Variables

A variable is the name given to the contents of a storage location, which can be changed while a parts file is running. There are 10000 of these locations available, each named with its location from 1 to 9999. Variables numbered 1 to 999 are persistent, meaning that their values are stored in the configuration file upon exiting TurboCNC and re-loaded when TurboCNC is restarted. Variables with names from #1000 through #9999 are transient. These values are not restored when TurboCNC is restarted. Variables that are not set before being read are given a value of zero.

A value is assigned to a variable by use of the assignment operator (the '=' sign). The assignment may appear anywhere a word can be placed within the block. For example:

```
G80 #7 = 2 F10   ; assignment does not have to be
                 ; first in a block
#7 = 4.25        ; assigns the value '4.25' to variable #7
```

A variable may be substituted for a value in all but the 'G', 'M', and 'T' code words. The following are legal blocks of code:

```
G00 x#7          ; Moves the x axis to 4.25 (continuing
                 ; from above)
G00 x[#7 - 4.25] ; Moves the x axis to 0 (expressions
                 ; resolved first)
```

A variable's name may be given as the contents of another variable, or as an expression.

```
#1 = 2.05      ; Assign the value 2.05 to variable #1
#2 = 1           ; Assign the value 1 to variable #2
G00 x##2       ; Move the x axis to 2.05 (##2 -> #1)
G00 x#[3 - #2] ; Move the x axis to 1 (3 - 1 = 2, #2 = 1)
```

Recursion of variable names has been tested to 3 levels (ie: ####1)

NOTE: Variables are not allowed as operands for 'G', 'M', 'N' or 'T' OpCodes.

Sample Code - Using Expressions and Variables

This code doesn't do anything especially useful, but it demonstrates some of the valid use of expressions and variables. It is taken from the parser test programs.

```
F[1+2]            ; Addition
F[1+2*4-3]        ; Brackets used to modify
F[(1+2)*(4-3)]    ;  order of operations

F[LN(10)]         ; Natural Log (e)
F[EXP(2.302585)] ; e^n

F[SIN(45)]        ; Sin (Trigonometric functions work)
F[ARCSIN(.707)]  ; Arcsin    (in decimal degrees)

#0 = [10-10]      ; Set variables 0 to 4
#1 = [10-9]       ;  with values equaling
#2 = [8/4]        ;  their positions
#3 = [SQRT(9)]    ;
#4 = [2*2]        ;

#5 = [#2+3]       ; Set variables 5 to 9
#6 = [#5+1]       ;  with values equaling
#7 = [SQRT((#4+#3)^2)]  ;  their positions
#8 = [#3^#2-#1]  ;
#9 = [SQR(#3)]    ;

( Wild ones )
#0=1 #1=2 #2=3 #3=4 #4=5 #5=99.99 ; Set variables
#6=50 #7=10 #8=0
F1                ; Feed rate = 1
F#[#2+#3]         ; Feed rate = 10
F##3              ; Feed rate = 5
F##[####[#7-10] - 2]    ; Feed rate = 4

#1=0 #2=0 #3=0 #4=0 #5=0   ; Clear variables
#6=0 #7=0 #8=0 #9=0 #10=0
#999=0
m02               ; end program
```

Conditional Execution (IF)

Function: The IF statement allows code to be executed if a condition is met. One good use of this statement is to cut the same contour out of material with varying thickness in multiple passes. Passes are made until an end condition is reached.

<u>Syntax</u>: IF condition [code]

IF = keyword
condition = argument comparator argument
argument = value | variable | expression
comparator = EQ | LT | LE | GT | GE | NE
code = DAK RS-274 D

<u>Action</u>:

- If the condition is met, the remainder of the block will be executed.

- If the condition is not met the remainder of the block is skipped.

Example:

```
#1=.500                ; Material thickness
#2=.125                ; Depth of cut per pass
G00 Z2.0               ; Clear Clamps
G00 X0 Y0              ; Move to start coordinate
#3 = #1 F5             ; Set first cut depth & Feed rate
N0100 #3=[#3-#2]       ; Start of loop, compute new depth
G01 Z#3                ; Set new depth
N0200 M98 O1000        ; Jump to contour subroutine
IF #3 GT 0 M97 O0100   ; Jump to start of loop (if not done)
N0300 M02              ; Program End

; Contour cutting subroutine
N1000 G01 X1           ; 1" square
G01 Y1
G01 X0
G01 Y0
M99                    ; return from subroutine
```

WARNING !	This example is a simplistic view of a machining operation. Further checks to ensure that your tool does not descend into the table should be added.

Notes:

- Valid comparators are:
  EQ - equals
  GE - greater than or equal
  GT - greater than
  LE - less than or equal
  LT - less than
  NE - not equal

- Multiple if statements can be cascaded on a line i.e.:

```
M05 F1             ; Preset indicators
IF 2 EQ 3 F2 IF 2 EQ 3 M03; FALSE : FALSE Result=F1, Spindle OFF
IF 2 EQ 2 F2 IF 2 EQ 3 M03; TRUE : FALSE Result=F2, Spindle OFF
IF 2 EQ 3 F1 IF 2 EQ 2 M03; FALSE : TRUE Result=F2, Spindle OFF
IF 2 EQ 2 F1 IF 2 EQ 2 M03; TRUE : TRUE  Result=F1, Spindle ON - CW
```

Simulating Advanced Conditional Execution Structures

The implemented conditional execution statements allow the simulation of more advanced conditional structures such as:

- IF - THEN - ELSE
- REPEAT – UNTIL
- WHILE – WEND
- CASE OF – END CASE

IF - THEN – ELSE

Simulation of the IF - THEN – ELSE statement in its classic form requires a jump to the code to be executed if the condition is met. This should not be omitted, even if the code to be executed will fit within the current block, as the parser may execute the required jump to the remainder of the program before executing the desired code. (M-Codes are the first executed by the code sequencer).

```
N10000 IF #1 EQ M97 O10100   ; test for condition
N10010                       ; code for ELSE
N10090 M97 N11000              ; continue program

N10100                       ; code for condition met

N11000                       ; program continues here
```

REPEAT – UNTIL

```
N10000                       ; start of REPEAT LOOP

N10980 IF #1 EQ 1 M97 N11000 ; UNTIL condition test
N10990 M97 N10000              ; repeat loop, condition not
met

N11000                       ; program continues here
```

WHILE – WEND

```
N10000 IF #1 EQ 1 M97 N10020 ; test for condition
N10010 M97 N11000              ; exit loop if false
N10020                       ; code executed if condition met

N10990 M97 N10000              ; END of WHILE loop

N11000                       ; program continues here
```

CASE OF – END CASE

Simulation of the CASE statement uses an IF to est for each condition, followed by a jump to the code to be executed if the condition has been met (CASE is satisfied). The code to be executed if none of the CASES match may directly follow the last IF statement as shown , or a jump to the code may be used. The code for every CASE ends in a jump to the next line to be executed. Although this is not strictly required for the code of the final case, it's been included to help prevent bugs if more CASEs are added later.

```
N10000 IF #1 EQ 1 M97 O10100 ; first case
N10010 IF #1 EQ 2 M97 O10200 ; second case
N10020                       ; else

N10090 M97 O11000              ; continue program

N10100                       ; code for first case
N10190 M97 O11000              ; continue program

N10200                       ; code for second case
N10290 M97 O11000              ; continue program

N11000                       ; program continues here
```

These structures have been illustrated using a simple test for equality. Most of the examples could have been simplified by inverting the test, in other words testing for inequality in the examples. This has not been done so that an identical structure, with a readily apparent condition, can be used for every implementation,. The inverse conditions are given in the table below for those wishing to adopt that form as their standard:

comparator	NOT comparator
LT	GE
LE	GT
EQ	NE
GE	LT
GT	LE

Interacting with the Operator

TurboCNC provides two functions that a CNC program can use to interact with the operator.

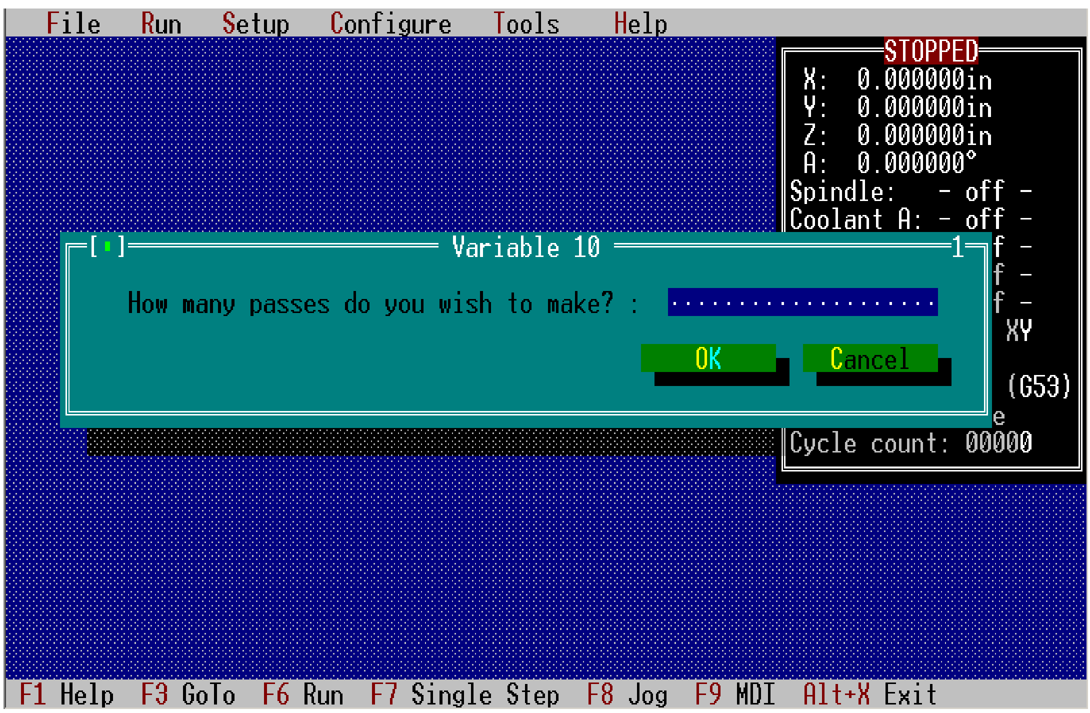
ASK

Function: Asks the operator for a value to be stored in a variable. An optional comment on the line will be used as a prompt.

Syntax: ASK #n ;Comment

Example:

```
ASK #10    ; How many passes do you wish to make?
```



Notes:

- The value entered can be used as any other variable during machining operations.

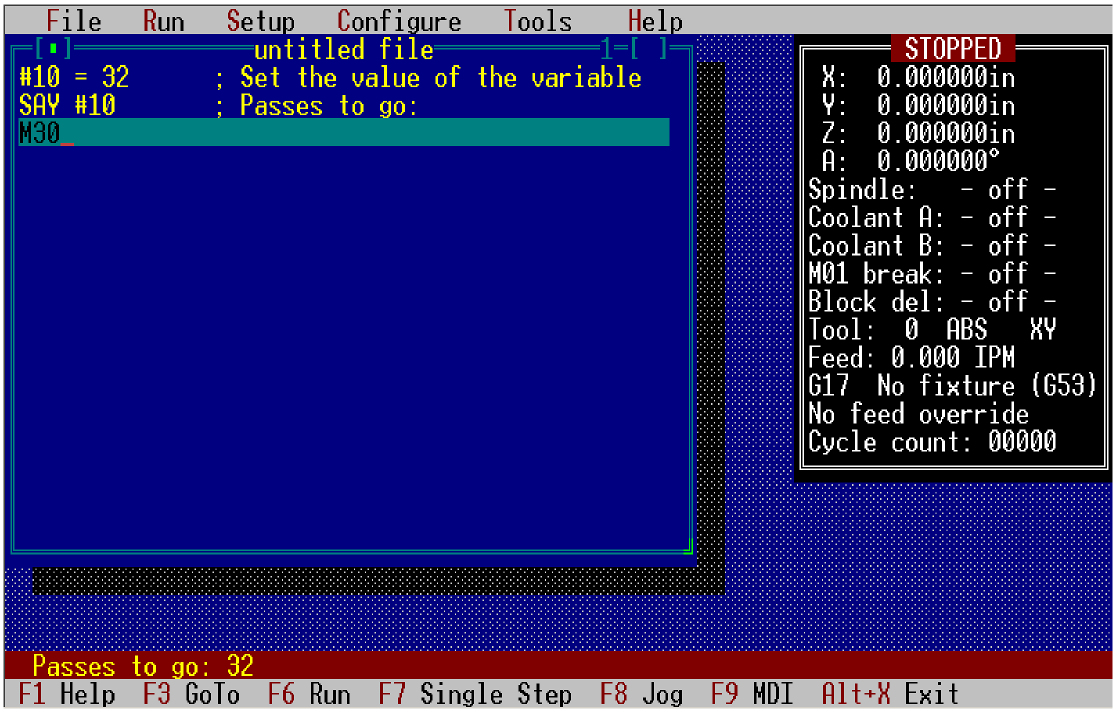
SAY

Function: Displays the content of a variable, and an optional comment on the bottom line of the screen.

Syntax: SAY #n ;Comment

Example:

```
#10 = 34  ; Set the value of the variable
SAY #10   ; Passes to go:
```



Notes:

- SAY will lockout the Port Monitor and clear the Port display (if active).
- SAY #0 will clear the screen
- SAY displays the content of the specified variable when invoked. The displayed value is not updated when the content of the variable is changed.
- Only one variable can be displayed on the screen. Subsequent calls to say will overwrite the initial display.

Putting it all Together: The Circle Using Line Segments

A program to mill a circle using line segments is used to illustrate best practices for the use of variables, expressions and conditional programming. While this isn't the most efficient method for cutting circles, it can be easily adapted to cut hexagons, octagons, or other regular polygons.

TurboCNC 'Interprets' the code on a line-by-line basis; it does not compile and optimize the result. It also scans the file to find the target of a jump function (M97). This takes time depending on how far into the program the jump's target is located. This is the reason the subroutines are located first. The subroutines should be ordered such that those executed most often are located first; then trailing down to those executed least often. The main part of the program, including initialization is therefore located last.

The reason variables #1000 and up were chosen for this example is that their values do not need to be preserved when TurboCNC exits. Always ensure that you initialize the appropriate variables prior to using them. Previous programs may have left a value that could produce disastrous results.

The Program:

```
; TurboCNC 4.0 program to cut circles
M97 O9000                  ;Jump to the main program


; *** Subroutine to cut the circle ***
N0100 #1005=[#1005+#1004] ;Increment the angle
SAY #1005                  ;Current angle (Stop at 360):
#1006=[#1000+(#1002/2)*COS(#1005)] ;compute new X position
#1007=[#1001+(#1002/2)*SIN(#1005)] ;compute new Y position
G01 X#1006 Y#1007          ;make the cut
IF #1005 LT 360 M97 O0100 ;Jump if circle not complete
M99                        ;return from subroutine


; ********************************
; *          Main Program        *
; ********************************
;Parameters for the circle (get the center from the operator)
N9000 ASK #1000      ; What is the X-coord of center
(inches):
ASK #1001            ; What is the Y-Coord of center
(inches):
#1002=1              ;Diameter of the circle (inches)
#1003=360            ;# of steps for a full circle

;computed variables
#1004=0              ;delta (angular increment)
#1005=0              ;theta (current angle)
#1006=0              ;next X position
#1007=0              ;next Y position

;setup
F20                  ;set feed rate
#1004=[360/#1003]    ;compute theta (TurboCNC 4.0
                     ; trig functions are in degrees)
#1006=[#1000+(#1002/2)*COS(#1005)] ;compute starting X
                                   ; position
#1007=[#1001+(#1002/2)*SIN(#1005)] ;compute starting Y
                                   ; position
G00 X#1006 Y#1007    ;traverse to start position
G01 Z2               ;lower Z-axis to make cut
N9100 M98 O0100      ;jump to cutting subroutine
G01 Z4               ;retract Z-Axis
SAY #0               ;clear display
M02                  ;End of program - circle.cnc
```

Part 4 – Introduction to CNC

General

If you're reading this section, you're probably a beginner to the CNC world. That's ok, we all start somewhere.

Chances are, you already understand machining. G & M codes provide a **formal language** for telling a computer how to do your work in the machine world.
When you make a part on a machine tool, say a lathe, and watch what you really do - you'll notice that a lot of the game is turning hand wheels to make very repetitive and exact motions.

CNC involves reducing the motions to codes in a scripting language that a computer can understand. In principle, if the computer can duplicate the same motions that a human does when machining, it duplicate the machining process. This is important to understand. G & M code **only spells out the motions involved in machining**. The computer doesn't know anything else, like where the part is, what kind of tool you're using, or even what kind of machine it's running. Motion only.

G & M code works on a line by line basis using words, left to right and top to bottom just as you are reading this page. A **word** has a special definition in this context however. In the RS 274D language, each word consists of a letter or symbol followed by a number. Here are two words as an simple example:

```
G00 X1.000
```

Each line of code is sometimes called a **block**. Let's break this block down and see what it means to the computer.

The first word "G00" (pronounced GEE-zero) is the **Rapid Positioning** word. It means move somewhere as fast as you can. Yes, those are zeroes after the G.

"'X1.000" is a word which gives a **new location for the X axis**; 1.000 in this case. If you've used a DRO equipped machine before, you probably understand coordinate systems. If not, have a look at the Axes De-mystified section and familiarize yourself with the concept.

When this block is executed by TurboCNC, one of three things will happen:

1. In Absolute Mode, the X axis will move to the 1.000 coordinate position as fast as possible

2. In Incremental Mode, the X axis will move +1.000 units as fast as possible

3. If there is no X axis to speak of, an error will be produced or the block will be ignored.

Not so bad, right? Just tell the machine what to do. By stringing a large series of blocks together you can create a script for the computer to follow. Each time you run the script (G code file), the computer performs the machining operation for you. Now run the script over and over, put in fresh stock each time, and you're in production!

Here are some general things you need to know about the language:

- It is standardized (pretty much, at least insofar as the basics are concerned). You can use it just about anywhere.
- G words usually do the actual motion.
- M words usually do miscellaneous functions.
- Spaces don't matter, but line breaks do.
- Read it left to right, top to bottom - just like you're reading this page.
- If the same word is used in a line as in the line before, it usually doesn't have to be retyped on the second line. This is called **Modality**. The old word is assumed until another one is used. Not all words are modal.
- The computer only does what you tell it to do, and will cheerfully snap a tool or drill through the table at your command.
- Words are also called "codes" in some circles.
- M02 is the word for "End of program" - often interchangeable with M30 which is "End of tape".

The Axes De-Mystified:

Any machine (lathe, mill, drill, etc) has its major linear axes defined in the standard orthogonal manner, that is at right angles to each other. The point to remember is that they are defined with respect to 'tool motion', not the actual motion of the axes.

Positive direction is found using the right-hand rule for a mill. Lay your right hand, palm up, on the table of the machine, thumb along the X-Axis, index finger along the Y-Axis, and the middle finger along the Z-Axis. Your fingers and thumb will point in the positive direction of movement for the tool along each axis. For a moving table machine such as a Sherline, positive movement along the X-Axis moves the table to the left, positive movement along the Y- Axis moves the table towards the front, away from the column. A gantry style machine moves the tool, so movement of the gantry will follow the right hand rule (exactly the opposite of the moving table).

By convention, rotational axes with their axis of rotation parallel to the X, Y, and Z axes, are labeled A, B, and C respectively. The direction of positive rotation of these axes is counterclockwise when viewed from the positive end of the corresponding orthogonal axis (X, Y, or Z). So, taking the Sherline as an example, with the rotary table mounted so that its axis of rotation is parallel to the X-Axis, with its table facing right, when viewed from the direction of the hand-crank (or stepper motor) rotates counterclockwise. In other words the top of the rotary table will turn towards the front of the mill. Something interesting to note is that the direction of rotation with respect to the surface of the rotary table will change if you turn it around to mount it facing left on the mill's table.

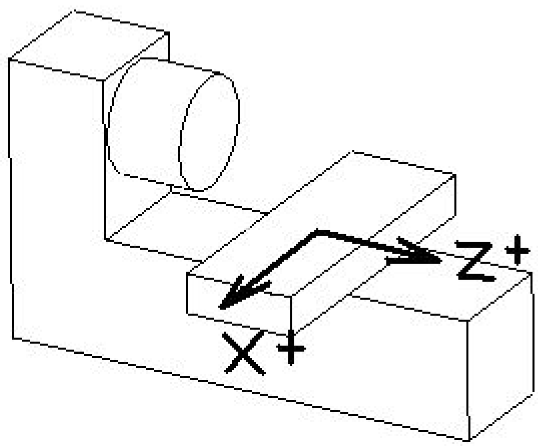
With only two axes, the right hand rule returns indeterminate results when used on a lathe. Machinists have adopted the convention that movement from the headstock to the tailstock is along the Z-Axis. Movement of the cross slide towards the tailstock is in the positive direction. In and out movement of the cross-slide is along the X-Axis. Movements of the cross slide, away from the centerline between the headstock and tailstock are in the positive direction.

The point of origin for the orthogonal axes can be set anywhere that is convenient. With the exception of a lathe's X-Axis, many machinists set this is at the point of maximum negative movement along the axes in the machine coordinate system. The machinist will switch to an alternate coordinate system for machining, and set a zero relative to the work piece. For the X-Axis of a lathe, by convention the centerline between the headstock and tailstock is chosen as zero.

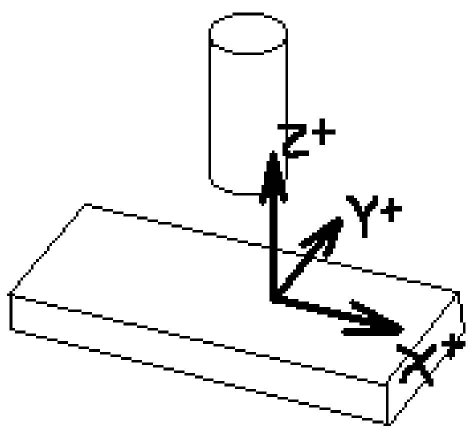
Axis conventions:

Per the Standard, X, Y and Z should always be linear; A, B, and C should always be angular. The XYZ coordinates should be perpendicular, with their directions arranged according to the right hand rule.

On a two-axis lathe, the axes should be named Z and X, arranged as shown below:



On a three axis mill, the axes should be named X, Y and Z - arranged as shown below. Be aware that this is tool motion relative to the work – typically the table will move opposite the diagram below.



The A, B, and C axis of rotation should be parallel to the X, Y, and Z-axes. U, V, and W should be linear also, and parallel to X, Y, and Z respectively.

Nothing will stop you from making X an angular axis in TurboCNC; in fact this may be the most expedient way to program an operation. In general you're better off going with the established practices though.

Arc directions (Clockwise vs. Counterclockwise) are explained under G02. The direction of an arc is taken by looking in the negative direction on the out of plane axis. For example, on a mill an arc in the XY plane (plane of the table) should look clockwise as looking in –Z (down from above).

The parallel port explained:

On most PC computers a 25 pin connector called the **parallel port** is available for connection to a printer or other hardware. By sheer accident of engineering, this port makes a great interface for controlling CNC machines.

The pins on the parallel port have two voltage levels. High is defined as 5V, low is 0V (Strictly speaking, 0-0.8V is low, 2.5-5V is high). This is known as TTL for Transistor-Transistor-Logic.

Communicating with an axis on a CNC machine is usually with two output pins. One is called the **direction** pin and sets the direction that a motor driving that axis is to turn. The other is the **step** pin, which is toggled each time the motor should rotate by a small amount.

On the parallel port, pins 2-9 are always available for output, which allows at least 4 axes of motion to be controlled. Pins 1,14,16, and 17 can be output also, to control spindles, coolant pumps, and tool changers.

The parallel port inputs are on pins 10,11,12,13, and 15. These are TTL level signals as well. Typical uses for these are spindle encoders, limit switches, and extra logic for detecting when you've run out of stock.

Pins 18-25 are ground. Use some of these to shield your cables.

TurboCNC can drive up to three parallel ports, at addresses $278, $378, and $3BC. Use the included FKEYBIT utility (a separate .exe) or the built in port monitor (F2 to activate) to test the lines on the port and provide signals for troubleshooting or proving out your setup.

Part 5 – Technical Details

The Parallel Port

Parallel Ports on later models of computer can be set to operate in any of the following three modes of operation:

- SPP (Standard Parallel Port - original specification)
- EPP (Enhanced Parallel Port)
- ECP (Enhanced Capabilities Port)

TurboCNC places the Parallel Port into the SPP mode for operation to ensure compatibility with the maximum number of computers.

The Standard Parallel Ports consists of three registers: the data, status, and control. The registers for first three ports are normally found at 0378h, 0278h, and 03BCh. These are the base addresses referred to below.

TurboCNC expects to find the ports at these addresses. If the computer has mapped these elsewhere, the addresses can be found in the BIOS at 0000:0408, 0000:040A, and 0000:040C for LPT1 through LTP3. The source of the registered version of TurboCNC can be modified to use the non-standard addresses.

Connections to the registers are as follows:

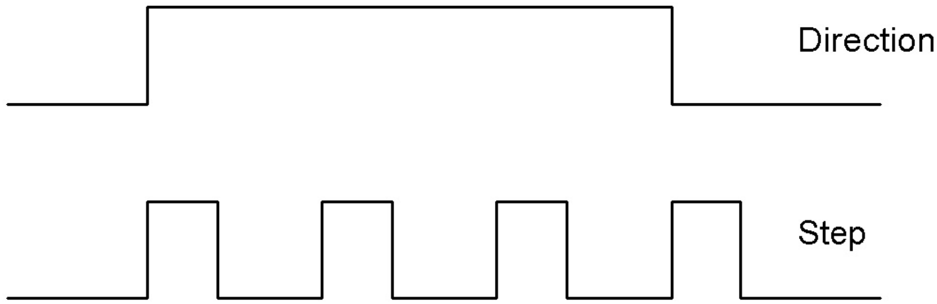
Base (Data)				**Base + 1 (Status)**				**Base + 2 (Control)**			
Bit	Pin	Dir	Purpose	Bit	Pin	Dir	Purpose	Bit	Pin	Dir	Purpose
7	9	Out	Data 7	7	11	In	*Busy	7			Unused
6	8	Out	Data 6	6	10	In	*Ack	6			Unused
5	7	Out	Data 5	5	12	In	Paper Out	5			Bi-Dir
4	6	Out	Data 4	4	13	In	*Select In	4			IRQ Enable
3	5	Out	Data 3	3	15	In	Error	3	17	I/O	*Select
2	4	Out	Data 2	2			*IRQ	2	16	I/O	Reset
1	3	Out	Data 1	1			Reserved	1	14	I/O	*Auto LF
0	2	Out	Data 0	0			Reserved	0	1	I/O	*Strobe

Notes:

- Pins 1, 11, 14, and 17 are inverted by port hardware
- A high logic level MUST be written to any bits of the control register used for input before reading it.

Set Up of Step and Direction Lines

Most drives and breakout boards are edge triggered. For example Gecko 201's step on the high to low transition. Our recommendation for this type of circuit is to use a normally low signal due to the method used in TurboCNC to generate the Step and Direction signals; like this



Assuming that +ve direction is +ve axis movement, and that the previous move has left the Direction line low, on writing the Step and direction signals to the port, both lines are switched high. There is no step taken... yet. This gives the Direction signal a period of time determined by that required by the CPU to execute a few instructions, and the PW delay set in TurboCNC to stabilize. This also gives the drive circuitry time to change gating logic as required to switch direction. After this period the Step signal is brought low and the step is taken. Note that there is no stabilization or gate switching period required for steps two or three. Step four does require these times, and it is provided with the setup shown. As a side note, this same setup should be used if the step is generated on the logic low. All that this does is delay the Step point by a few microseconds. Step to Step timing both for a single or multi-axis move is preserved.

For TurboCNC the Step and Direction settings in the .ini would be:

```
IsStep/Dir=True
PortAddress=$378
StepPin=2
IsActiveHigh=True
Pulsewidth=0
DirPin=3
LowIsPositive=False
```

For drives that step on the low to high transition or at the high logic level, IsActiveHigh should be set to False. This setup allows ringing on the direction line to die out, and provides the necessary time for the direction gates to settle before application of the Step pulse. This will prevent the lost steps which would otherwise occur during direction changes.

I/O Points

Point	Usage
Spindle Power	(Output) Drives relay that turns spindle motor on and off. Could also drive spindle brake.
Spindle Direction	(Output) Drives relay that sets spindle direction to clockwise or counter clockwise.
Coolant A	(Output) Drives relay that controls flood coolant pump
Coolant B	(Output) Drives relay that controls mist coolant pump
PLC Handshake Signal	(Output) Signal controlled by M70 and M71 to provide programmatic interface to a Programmable Logic Controller or other electronics
Drive Enable 1	(Output) Used to enable or disable one or more motor drives. Since some motor drives require individual enable lines, TurboCNC provides three independent lines which are controlled by the same M17/M18
Drive Enable 2	(Output) Used to enable or disable one or more motor drives. Since some motor drives require individual enable lines, TurboCNC provides three independent lines which are controlled by the same M17/M18
Drive Enable 3	(Output) Used to enable or disable one or more motor drives. Since some motor drives require individual enable lines, TurboCNC provides three independent lines which are controlled by the same M17/M18
Clamp Selector Bit 0	(Output) – used with Bits 1, 2, 3 to specify which clamp is to be controlled.
Clamp Selector Bit 1	(Output) – used with Bits 0, 2, 3 to specify which clamp is to be controlled.
Clamp Selector Bit 2	(Output) – used with Bits 0, 1, 3 to specify which clamp is to be controlled.
Clamp Selector Bit 3	(Output) – used with Bits 0, 1, 2 to specify which clamp is to be controlled.
Clamp Motor On Signal	(Output) drives clamp controller specified by Clamp Selector Bits 0-3.
Clamp Direction Closed	(Output)
Tool Turret Index	(Output) Toggles 1 times the tool number of pulses when an M06 instruction is executed.
Collet Open Solenoid	(Output) drives mechanism to open the tool holder, which may be a collet
Collet Close Solenoid	(Output) drives mechanism to close the tool holder, which may be a collet
Emergency Stop	(Input) when activated, this line causes the CNC machine to stop operation.
Limit Switch 1	(Input) Is activated when Axis #1 reaches either limit of its travel.
Limit Switch 2	(Input) Is activated when Axis #2 reaches either limit of its travel
Limit Switch 3	(Input) is activated when Axis #3 reaches either limit of its travel
PLC Handshake Sense	(Input) Used to synchronize the CNC machine to a Programmable Logic Controller or other electronic device.
Spindle Index	(Input) Used to synchronize lathe's lead screw to spindle for

	gearless thread cutting. Can also be used to calculate and display spindle speed.
Spindle Encoder A	(Input) Not currently used
Spindle Encoder B	(Input) Not currently used
Touch Probe	(Input) Stops motion on a G31/32 or probe move in jog mode. Often used for digitizing part data or toolsetting.
Jog Encoder A	(Input) Quadrature channel A of a jogging encoder wheel. In discrete jog mode, this is an input to move an axis.
Jog Encoder B	(Input) Quadrature channel B of a jogging encoder wheel. In discrete jog mode, this is an input to move an axis.
Block Hold	(Input) When active, TurboCNC will not execute the next block in the CNC program. Could be used as a single step control.
Start Inhibit	(Input) When active, TurboCNC will not start executing a CNC program. Could be used to allow a raw materials handler to remove completed part from machine and insert new raw material.
Clamp Sense Opened	(Input) line which is activated when the clamp specified by the Clamp Select bits is opened
Clamp Sense Closed	(Input) line which is activated when the clamp specified by the Clamp Select bits is closed
Home Switch 1	(Input) Senses when Axis #1 is at home position
Home Switch 2	(Input) Senses when Axis #2 is at home position
Home Switch 3	(Input) Senses when Axis #3 is at home position
Home Switch 4	(Input) Senses when Axis #4 is at home position
Home Switch 5	(Input) Senses when Axis #5 is at home position
Home Switch 6	(Input) Senses when Axis #6 is at home position
Home Switch 7	(Input) Senses when Axis #7 is at home position
Home Switch 8	(Input) Senses when Axis #8 is at home position

Configuring Speed Control

To be deemed valid, a map must contain at least two mappings. A maximum of 1023 mappings may be specified. Mappings may be entered in any order, but when sorted both the count values and RPM must increase from mapping to mapping.

A quick method of determining the mappings for the Speed Control is to create a map with a 1:1 relationship of count Values to spindle RPM. The count values can then be directly entered as a spindle speed, and the actual speed measured with a tachometer. The mappings should be noted and entered as a separate map upon completion.

High and low limits are recomputed for the selected map every time a mapping is added. This can occur while reading a configuration file or adding entries using the configuration menus. Clearing the map sets the RPM limits to default values of 100,000,000 for the low end and 0 for the high end.

Setting up Windows 9x to boot directly into MS-DOS

Although Microsoft has advertised Windows 95, 98, and 98-Second Edition as having a multitasking “native mode,” these operating environments still include a version of MS-DOS and can be configured to boot directly into MS-DOS.

Modify MSDOS.SYS

If you are running in the windows environment, click on the “start button” in the lower left corner of the screen. From the menu that appears, select “RUN”

In the dialog box that appears, type

attrib MSDOS.SYS –r –s –h

Next, use notpad or some other ascii text editor to open MSDOS.SYS in an edit window.

MSDOS.SYS is formatted like an initialization file, with a series of sections (e.g. [Options] or [Paths]) followed by a list of variables and the items to which they are assigned.

Find the line that reads

BootGUI=1

Change it to read

BootGUI=0

This enables your computer to boot directly into DOS. Save MSDOS.SYS.

Press the “Start” button and select “RUN.”

Enter the command

attrib MSDOS.SYS +r +s +h

Modifying CONFIG.SYS

You can now modify a file named **CONFIG.SYS** to display a menu which allows you either to boot directly into MS-DOS or into the Windows environment.

Press the “Start” button and select “RUN.”

Enter the command

attrib CONFIG.SYS -r -s –h

Next, open CONFIG.SYS withNotepad.

Some Windows 9x computers may not use a copy of CONFIG.SYS. If yours does not, use Notepad to create it.

At the beginning of CONFIG.SYS, add the following lines:

MenuItem=WIN, Windows GUI
MenuItem=DOS, MS-DOS 7.1 Unadorned
MenuDefault=DOS, 10
[WIN]

….. The remaining entries of your CONFIG.SYS file are here

At the end of your CONFIG.SYS file add the following lines:

```
[DOS]
DeviceHigh=C:\windows\command\ansi.sys
Shell=C:\COMMAND.COM C:\ /E:3072 /P
Files=50
```

This configuration file will display a menu with two items. It will wait ten seconds, and then default to the “DOS” selection. During the ten second wait, you can use the cursor keys to select the Windows GUI instead. You can also set up CONFIG.SYS to select Windows as its default. If you want to do this, change the line

MenuDefault=DOS, 10

to

MenuDefault=WIN, 10

Save CONFIG.SYS

Press the “Start” button and select RUN.

Enter the command

attrib CONFIG.SYS +r

Modify AutoEXEC.BAT

Open the Autoexec.bat file using notepad

At the end of the file, add the following lines

```
IF “%CONFIG%”==”WIN” C:\Windows\Win.COM
IF “%CONFIG%”==”DOS” C:\DOS\MOUSE.COM
```

Save autoexec.bat and exit notepad.

Reboot your system. You should see a menu, with a selection highlighted. If DOS is selected, you’ll boot into DOS and can execute TurboCNC.

Assuming that you have a folder in your root directory named DOS and that the folder contains the executable MOUSE.COM, the last line will load the mouse driver so you can use a mouse or track ball to access TurboCNC’s menu structure.

1445 Koll Circle #104
San Jose, CA 95112
(408) 436-7777
FAX (408) 436-7910

Please read the terms of this agreement before opening this package. If you open the diskette package or keep it for more than 30 days, you accept the agreement completely.

Program License Agreement

1. License. BOBCAD-CAM grants a license to the user, individual and/or corporate, to use the Program Product for a term beginning on the date you accept this agreement and continuing until termination. This license agreement is automatically terminated if you violate any of the terms and conditions of this license agreement.

2. Use of the Program Product. You may use the Program Product on any one IBM personal computer or functional equivalent. You do not have the right to make unlimited copies of this software as it is protected by computer software copyright laws. Backup copies are for personal use by the purchaser only, and any additional copies are unlawful without written permission from BOBCAD-CAM. It is illegal to use this product on more than one machine at a time.

3. Restrictions. You may not use, copy, modify, merge, or transfer the Program Product or any portion or modification thereof in whole or in part, except as expressly provided in this agreement.

4. Limited Warranty. BOBCAD-CAM warrants the distribution diskette upon which the Program Product is provided to be free from defects in materials and workmanship under normal use for a period of 60 days from the date of purchase by the original purchaser. FOR BREACH OF ANY WRITTEN OR IMPLIED WARRANTY ON THIS PRODUCT, THE CONSUMER IS LIMITED TO THE REPLACEMENT OF THE DEFECTIVE DISKETTE.

5. General terms. This is the only agreement between User and BOBCAD-CAM regarding this Program Product and it may be modified only by a written agreement between those parties. This agreeement cannot be modified by purchase orders, advertising or other representation by any person. If any provision of this agreement shall be rendered invalid, illegal, or unenforceable, then the validity, legality, and enforceability of the remaining provisions shall not be affected or impaired thereby. This agreement shall be governed by the laws of the state of California. By opening or keeping the diskette package, you acknowledge that you have read this agreement, understand it and agree to be bound by its terms and conditions.

BOBCAD LICENSE AGREEMENT

ACKNOWLEDGEMENT
Graphics are generated with the use of GRAFMATIC (a Scientific/Engineering Graphics package), PLOTMATIC, and PRINT MATIC, all from MICROCOM-PATIBLES, 301 Prelude Drive, Silver Spring, MD 20901.

TABLE OF CONTENTS

SETTING UP BOBCAD

Transfer BOBCAD from the floppy disks to your hard disk as follows:

Using Drive A:
Insert the floppy disk into drive A.
Type: A:SETUP

Using Drive B:
Insert the floppy disk into drive B
Type B:SETUP-B

(The program SETUP.BAT will now automatically tranfer the files from the floppy disk onto your hard disk).

You are now in directory BOBCAD (or BOBWIRE) on your hard-disk. Copy any remaining diskettes (PRINTER DRIVERS and or RS-232 communications) using the COPY command as follows: (be sure to use B: instead of A: if you are using drive B)

COPY A:*.* C:\BOBCAD

or for BOBCAD WIRE type:

COPY A:*.* C:\BOBWIRE

EXCEPTIONS:

(If you do not use 'SETUP.BAT' you must observe the following: all files must be copied from the floppy disks onto your hard-disk into one directory except for the *.HLP files! The *.HLP files need to be in directory 'DHELP'. The 'DHELP' directory must be a subdirectory in the directory in which BOBCAD exists).

408 436 7777

HELP: If you need help call our support number: [redacted]24.
SALES OFFICE: (408) 436-7777.

STARTING BOBCAD

MOUSE USERS:
Your mouse must be initialized when you turn on your computer (before you start BOBCAD). This is described in the manual that came with your mouse.

DIGITIZER USERS:
If you have a Kurta digitizer, you must initialize it every time you turn on your computer. This is described in the manual that came with the digitizer. The Kurta must be initialized to high-resolution.

1. To start BOBCAD go to directory BOBCAD as follows:

   Milling users:
   Type: CD \BOBCAD (and hit the Enter or Return Key).

   Wire EDM users:
   Type: CD \BOBWIRE (and hit the Enter or Return Key).

2. Next type BOBCAD.

3. If this is your first drawing, select 'START A NEW DRAWING' from the start menu (Press 1).

4. Next type in any name of up to 8 characters. (Or longer if you add a directory to the name).

5. Do the beginning excercises in this manual and read the NOTICE TO ALL USERS on the next page.

PRINTER INFORMATION:
(For detailed information see the PRINTER SECTION in the back of the manual.)
If you run DOS 5.0 and have a VGA card then type GRAPHICS after you have turned on your computer. Then when you have a drawing on the screen you can press the PRINT-SCREEN key on your keyboard. This will print the drawing on your printer. (Not available for Laser or Ink-jet printers). The color of the Entities need to be white for this to work well. You can put the GRAPHICS command in your AUTOEXEC.BAT file so it will load automatically.
For a more accurate plot use the PRINTDGN routine as described in the back of this manual.

NOTICE TO ALL USERS

If you are like most readers and do not read the manual, please read the following points:

1. You should do the beginning exercises in the manual and the more advanced ones to fully get to know all the features of **BOBCAD**.

2. After you have made a few drawings, look at the pages called **QUICK KEYS**. The Quick Keys are very handy and will make your work go faster. It will also speed things up if you look at the Overlay Strip for your keyboard.

3. Press **H** for **Help** when a menu is on the screen. Press any key to turn **Help** off.

HARDWARE REQUIREMENTS

BOBCAD runs on IBM* PC compatible computers: XT, AT, 386 and 486 series.

*Trademark of IBM

BOBCAD supports CGA, EGA and VGA graphics cards.

COMPUTER

The following configurations are typical examples. The Math. Co-Processor is optional, but does make BOBCAD 5 to 10x faster! With 640k of RAM you can only draw 1000 elements. An additional 2 MEG of RAM is required to be able to draw 11,000 elements.

PC-XT 640K, 20MB Hard Disk, 360K Floppy Drive, Math Co-Processor (be sure to consider that BOBCAD takes up 580K, your mouse takes up about 21K, so your DOS program must take up less than 40K).

PC-AT, 2 MEG RAM, 44MB Hard Disk, 1.2 MB Floppy Drive, Math. Co-Processor

386 Series, 4 MEG RAM, 80MB Hard Disk, 1.2MB Floppy Drive, Math. Co-Processor

486 Series, 4 MEG RAM, 120MB Hard Disk, 1.2MB Floppy, Math. Co-Processor

GRAPHICS CARD + MONITOR

CGA card + monitor. Note: BOBCAD supports this card in 2 color mode only (for example Black and White). There will be no future development for this card.

or EGA Colorcard 640x350 resolution and color monitor

or VGA Colorcard 640x480 resolution and color monitor

INPUT DEVICES

Mouse (Microsoft compatible)

Or Digitizing Tablet (Summagraphics compatible) or KURTA XLC (large tablets).

Scanners can be used to scan a drawing or sketch. DRAFTSMAN Is needed to transfer a scanned image into BOBCAD. For example our customers are using a LOGITECH hand-held scanner to scan a sketch. The information is then run through DRAFTSMAN and then into BOBCAD. DRAFTSMAN is optional from BOBCAD-CAM.

OPTIONAL OUTPUT DEVICES

Dot Matrix Printer (can print drawing on paper)

Or Plotter (HPGL compatible)

Or Laser Printer or Desk Jet

Configuring BOBCAD

To configure BOBCAD return to the DOS prompt and run the program CADCFG. This program updates the configuration file which is called CAD.CFG.

The following can be set:

Graphics card: CGA / EGA / VGA
Aspect ratio (to make sure circles look round)
Color of screen, menu, entities (entities must be white for the Print-screen key to work well).
Start up scale and location of X-Y axis
X-Y axis display
Mouse or tablet (9x6, 12x12, 12x18 or KURTA XLC)
Speed of mouse
Automatic save interval
Plotter port and plotter baud rate
Chain gap (Sets the gap that the chain select functions and the SINGLE and AUTO functions in the MILLING MAIN MENU will jump across).

To configure the CAM output of BOBCAD refer to the section NC CONTROLLER : HOW TO CONFIGURE BOBCAD.

CAD EXERCISES

Section2

This section contains the following excercises:

2.1) GET TO KNOW BOBCAD: BASIC EXERCISES.

2.2) DRAWING A RECTANGLE WITH ROUND CORNERS.

2.3) COPY/MOVE EXERCISES.

2.4) MIRROR EXERCISES.

2.5) ARC TOUCHING 3 ENTITIES.

2.6) DRAWING A PART (INTERMEDIATE).

2.7) ADVANCED EXERCISE.

GETTING TO KNOW BOBCAD

YOUR FIRST EXERCISE!

In this exercise you will learn how to draw Points, Lines, Arcs and Circles.

NOTE: Every 20-30 minutes BOBCAD will automatically ask you if you want to save your drawing. You have the option of either Yes or No.

Start with the first three functions in the **POINT MENU** as follows:.

First make sure you are in the **MAIN MENU**: Read the line on top of your screen, (if it does not say **MAIN MENU** hit the 'M' key on your keyboard and you will return to the **MAIN MENU**).

From the **MAIN MENU** select the function **POINT** as follows:

With the mouse move the HI-LIGHT to the function **POINT** and then click the left button on the mouse to select this function. You will now be in the **POINT MENU.**

The **POINT MENU** looks like this:

POINT MENU
1. **SKETCH**
2. **COORDINATES**
3. **INC. X,Y**
4. **END**
5. **INTERSECT**
6. **ARCCENTER**
7. **PERPENDICULAR**
8. **ON ENTITY**

Now, with the mouse move the HI-LIGHT bar in the menu to function 1. **SKETCH** .
Next, click the left button on the mouse.
Result : the cursor comes on your screen.

With the mouse move the cursor to any position you want.
Next, click the left button on your mouse.
Result: A Point will is drawn on your screen (shown as an 'x').

Again move the cursor with the mouse.
Click the left button on the mouse and another Point is drawn.

To get out of this fuction click the right button on the mouse or hit the **Esc** (**Escape** key) on your keyboard.

Well? Was that simple? (At any time, if you run into difficulties, give us a call).

So remember: to select a function, move the HI-LIGHT bar to the function, then click the left button on the mouse.

DRAWING A POINT USING COORDINATES

Select function 2. **COORDINATE** from the **POINT MENU**.
Now type in **2** for the **X** value and hit the **Enter** key.
Now type in **2** for the **Y** value and hit the **Enter** key.
Result: A point is drawn at coordiates X=2, Y=2

Now type in **4** for the **X** value and hit the **Enter** key.
Now type in **4** for the **Y** value and hit the **Enter** key.
Result: A Point appears at coordinates X=4, Y=4

Next, click the right button on the mouse to go back to the **POINT MENU** (or hit the **Esc** key on your keyboard).

HELP

Now that you are back in the **POINT MENU** hit the H key on your keyboard. The help page will appear for the **POINT MENU**. Note: Each menu has a help page like this. To turn Help off, select any function or hit the 'H' key.

Function 3.INCREMENT

We will now draw a Point using the **INCREMENT FUNCTION** as follows:

Select function 3. **INC X Y (INCREMENT)** from the **POINT MENU**.
The cursor should be back on the screen.
With the mouse move the cursor to one of the Points on the screen.
Next, click the left button on the mouse to pick this Point which will turn red.
Now type in -2 for the X value and hit the Enter key.
And type in 0 for the Y value and hit the Enter key.
Result: A new Point is drawn 2 inches to the left of the Point that you picked.

The cursor should still be on the screen.
With the mouse move the cursor to another Point.
Click the left button on the mouse to pick this Point.
The menu still shows the last values that you entered.
Now click the left button on your mouse or hit the Enter key to accept the X = -2 value.
Next type in 2 for the Y value and hit the Enter key.
Result: A new Point is drawn 2 inches left and 2 inches up from the Point that you picked.

Click the right button on the mouse to return to the **POINT MENU**.

Click the right button on the mouse again to return to the **MAIN MENU**.

You should now have an idea of how you control **BOBCAD** through the mouse. Remember the Esc key on the keyboard gets you out of any function. The M key also gets you back to the **MAIN MENU**.

****

This leads us to the following very important information: The **M** key is called a **QUICK KEY**. Many of the other keys are also **QUICK KEYS**. For example, when you hit the "L" key you instantly go to the **LINE MENU**.... FROM ANY OTHER MENU.

To find out which are the **QUICK KEYS** look at the colored letters in the **MAIN MENU**. Try this now by hitting the **P** key, now the L key, now the T key, now the **Z** key, etc.. you will notice that you go from menu to menu.

****

We will now continue and work with some of the Line drawing functions.

Hit the **L** key on your keyboard to go to the **LINE MENU**. The following menu appears:

LINE MENU
1. **SKETCH**
2. **COORDINATE**
3. **JOIN**
4. **TANGENT**
5. **PARALLEL**
6. **ANGLE**
7. **CHAMFER**
8. **CONTINUOUS**

Function 1. SKETCH

Now select function 1. **SKETCH** from this **LINE MENU** by hitting the number 1 on your keyboard (As you can see you can also select a function by hitting the number in front of it. You do not have to use the mouse to select a function).
The cursor should now be on the screen.

With the mouse move the cursor to where you want the start of the Line to be and click the left button.
Next, move the mouse to where you want the end of the Line to be and click the left button again.
***Result:* A line is drawn on the screen....**

....BUT....YOU CAN NOW TURN THIS LINE INTO A PERFECTLY HORIZONTAL OR VERTICAL LINE by typing in an X or Y coordinate.

So let's type in a coordinate, for example: 2 , and you see that the Line snaps to horizontal or vertical (depending on the angle you drew it at) through the coordinate that you entered.

If you do not type in a coordinate but hit the right button on the mouse the Line will stay where it is.

Try drawing a few more Lines in this manner.

Note: The SKETCH function that you just used is a free-hand way of drawing Lines. Do not try to use it to draw a Line from one existing Point on the screen to another Point. It will not be accurate! To accurately connect two Points you must use the JOIN function. We will try the JOIN function later in this exercise.

Now, return to the **LINE MENU**

<u>Function **2. COORDINATE**</u>

Select function **2. COORDINATE** from the **LINE MENU.**
Type in 1 for the **XS** value (start of the Line) and hit the **Enter** key.
Type in 1 for the **YS** value and hit the **Enter** key.
Type in 5 for the **XE** value (end of the Line) and hit the **Enter** key.
Type in 5 for the **YE** value and hit the **Enter** key.

Result:* *A 45 degree Line is drawn from 1, 1 to 5, 5.

Now click the right button on the mouse to exit the coordinate input menu and return to the **LINE MENU.**

Function 3. JOIN

Select Function 3. JOIN from the LINE MENU.

After you have selected the **JOIN** function the cursor will now be on the screen.
With the mouse move the cursor to one of the Points on the screen.
Next click the left button on the mouse to pick the Point. (The Point will become red).
Now move the cursor to another Point that is not on a Line and click the left button on the mouse.
Result: A Line is drawn that accurately joins the two points.

The cursor is still on the screen.
With the mouse move the cursor to the end of one of the Lines on the screen and click the left button on the mouse. (The Line will turn red).
Now move the cursor to the end of another Line and click the left button on the mouse.
Result: A new Line is drawn joining the ends of the Lines.

NOTE: With the 3.JOIN function that we just tried you can draw Lines that join Points, the ends of Lines or Arcs (or any combination of these). Because a Line or Arc has two ends, it is important that you move the cursor close to the end you want joined!

Now hit the **Esc** key or click the right button on the mouse to go back to the **LINE MENU.**

Function 5. PARALLEL

Now select function **5. PARALLEL** from the **LINE MENU.**

After you have selected this function the cursor will again be on the screen.

With the mouse, move the cursor to one of the Lines on the screen and click the left button to pick this Line (which will turn red).
Now move the cursor to the side of the Line where you want the new Line and click the left button.
Type in **.5** for the distance and hit the **Enter** key.

Result: A Line is drawn that is parallel to the Line you picked at a distance of 0.5.

Now click the right button on the mouse or hit the **Esc** key on your keyboard to return to the **LINE MENU.**

DRAWING ARCS AND CIRCLES

Hit the 'A' key on your keyboard to go instantly to the **ARC MENU.**

The **ARC MENU** appears as follows:

ARC MENU
1. **SKTCH CNTR**
2. **COOR CNTR**
3. **POINT CNTR**
4. **FILLET**
5. **3 ENTITIES**
6. **CHAIN**
7. **ELLIPSE**

Function 1. SKETCH CENTER

Select function **1.SKTCH CNTR** from the **ARC MENU**

Now type in 1 for the radius and hit the Enter key.

Hit the Enter key again to accept the default value for the Start Angle.

Hit the Enter key again to accept the default value for the End Angle.

A circle appears on the screen.

Use the mouse to move the circle to where you want it and click the left button on the mouse.
Result: A full Circle is drawn around the screen position you picked.

As you can see the value menu is back on the screen.

To accept the old value of a radius of 1 just hit the Enter key (or click the left button on the mouse).
Again hit the Enter key to accept 0 for the Start Angle.
Type in 135 for the End Angle and hit the Enter key.
Next move the 135 degree Arc with the mouse to where you want it and then click the left button on your mouse to pick that position.
Result: an Arc is drawn around the screen position you picked.

Note: In BOBCAD like in most CAD systems Arcs and Circles are drawn counter-clockwise around the center of the Arc. 0 degrees is in the positive X-axis direction (3 o'clock position), 90 degrees is in the positive Y-axis direction (12 o'clock position) etc. This rule does not affect the CAM or NC output.

BOBCAD does not make a difference between an Arc and a Circle. In most cases the Circle starts at 0 degrees and then ends at 360 degrees. The start and end point of a Circle are the same. You must remember that full circles are always broken at the 0 degrees position.

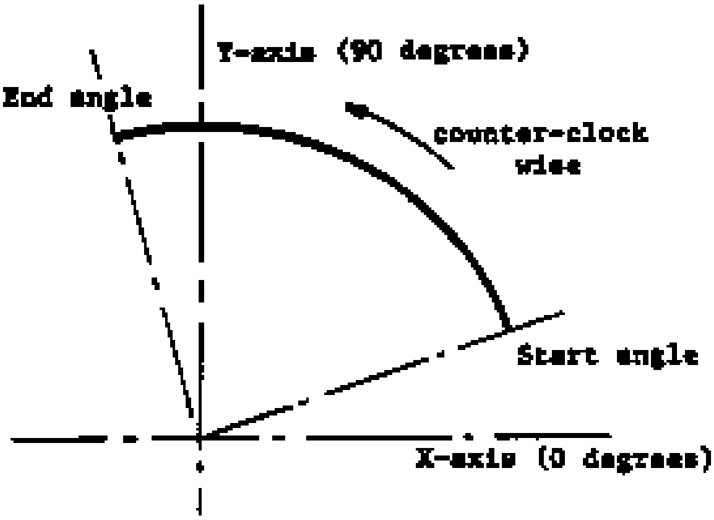


Figure shows how Arcs and Circles are stored in BOBCAD.

Now return to the ARC MENU by clicking the right button on the mouse.

Function 2. COOR CNTR (Coordinate Center)

Select function 2. COOR CNTR from the ARC MENU.

Type in 6 and hit the Enter key for the X-value of the center of the Arc.

Type in 1 and hit the Enter key for the Y-value of the center of the Arc.

Type in 1 for the radius and hit the Enter key.

Type in 0 for the Start angle and hit the Enter key.

Type in **180** for the End angle and hit the **Enter** key.
Result: an Arc is drawn with center 6,1.

Hit the right button on the mouse to return to the **ARC MENU**.

Function 3. POINT CNTR (Point Center)

Select function 3. **POINT CNTR** from the **ARC MENU**

Click the left button on the mouse to accept the default of 1 for the radius.

Click the left button again to accept the default value for the Start Angle.

Click the left button again to accept the default value for the End Angle.

The cursor is now on the screen.

With the mouse move the cursor to a Point on the screen and click the left button on the mouse to pick this Point (the Point will now be red).

Result: A full circle is drawn around the Point.

As you can see the value menu is back on the screen.

Type in **2** for the radius.

Type in **90** for the Start Angle.

Type in **270** for the End Angle.

The cursor is back on the screen.

With the mouse pick another Point.

Result: An Arc is drawn around the Point you picked.

That's great - you finished the beginning exercises!

For more information on all the menus and functions, look through the section in this manual called **MENUS**.

Now your screen may be cluttered with Entities (Entites are Points, Lines, Arcs, Dimensions etc.). So let's get rid of them before we go on with the next exercise.

Return to the **MAIN MENU** and select the **EDIT FUNCTION**.

You will see the following menu:

EDIT MENU
1. **DELETE**
2. **DELETE LAST**
3. **BLANK**
4. **UNBLANK**
5. **TRIM**
6. **BREAK**

From the **EDIT MENU** select function **1. DELETE**

You will now see the following menu:

DELETE MENU
1. **SINGLE**
2. **CHAIN**
3. **REGION**
4. **LAYER**
5. **ALL**

Select function **5.ALL** from the **DELETE MENU**. You will be asked the question **"DELETE ALL?"** with choices **1.YES , 2.NO** Choose **1.YES.**
Result: Everything that was on the screen is deleted.

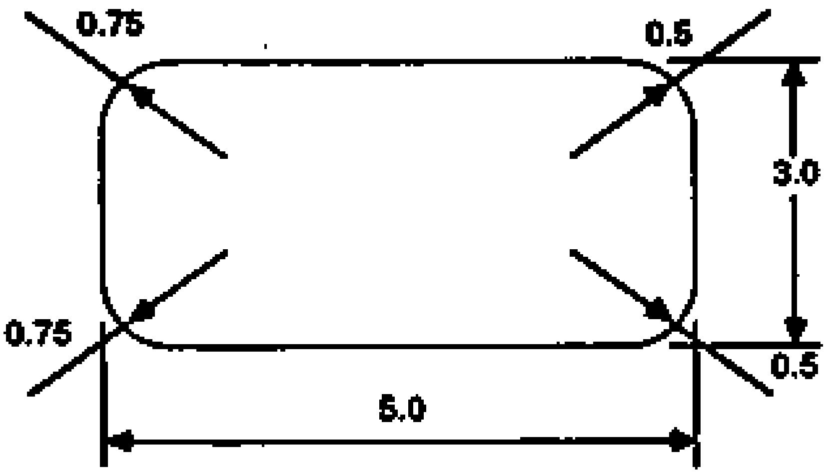
YOU ARE DONE WITH THE BASIC EXERCISES. YOU MAY WANT TO DO THESE TWICE SO THAT YOU BECOME COMFORTABLE WITH HOW BOBCAD USES THE MOUSE AND KEYBOARD.

First take a coffee break or do something else before we go on!

Drawing a Rectangle with Round Corners

We advice you to do this exercise. It covers many functional

5" x 3" RECTANGLE WITH ROUND CORNERS



Note: If you make a mistake press M. This allows you to start again from the MAIN MENU.

Note: You can Quit at any time by hitting the Q key on your keyboard. You will then exit BOBCAD by choosing one of the two quit functions.

Begin the exercise: (Refer to the figure shown above)

STEP 1: Draw the bottom Line as follows:
From the **MAIN MENU** select function **LINE**.
From the **LINE MENU** select function **2.Coordinate.**
Enter the coordinates shown below:

XS-COORD. = 2
YS-COORD. = 2
XE-COORD. = 7
YE-COORD. = 2

Result: The bottom Line is drawn on the screen (refer to the above figure).

Hit the **L** key to return to the **LINE MENU**

STEP 2:
From the **LINE MENU** select function **5. PARALLEL.**

The cursor should now be on the screen.

With the mouse, move the cursor to the Line you have just drawn.

Click the left button on the mouse to pick this Line (If you do not have a mouse, move the cursor with the arrow keys. Then use the Space bar on the bottom of the keyboard to pick the position.

The Line you have picked will now be red.

Next: Move the cursor above the Line to indicate on which side you want the Parallel line to be and click the left button on the mouse to indicate this position.

Type 3 for the Distance and hit the Enter key.

Result: The Top Line is drawn parallel to the Bottom Line.

Hit the Esc key or the right button on the mouse to return to the **LINE MENU.**

STEP 3:
From the **LINE MENU** select function **3. JOIN**

The cursor should now be on the screen.

Move the cursor near the left side of the Top Line and then click the left button on the mouse to select this end of the Line.

Now move the cursor close to the left side of the Bottom Line and click the left button on the mouse to select this end of the Line.

Result: A line is drawn that joins the Top and Bottom Lines on the left.

Draw the Line on the right in the same way!

After you have drawn the Lines of the rectangle hit the Esc key on the keyboard or the right button on the mouse to return to the **LINE MENU.**

Hit the **M** key to return to the **MAIN MENU.**

STEP 4: ZOOM

From the **MAIN MENU**, select the Zoom function.

You will now see the following menu:

ZOOM MENU
1. **WINDOW**
2. **NEW SCALE**
3. **VIEW ALL**
4. **SHIFT**
5. **PREVIOUS**
6. **REDUCE**
7. **3-D VIEWS**

From the **ZOOM MENU** select function **1.WINDOW**

The cursor is now on the screen.

With the mouse, move the cursor about 1 inch to the left and below the lower left corner of the rectangle. Click the left button on the mouse.

Now move the cursor about 1 inch to the right and above the upper right hand corner of the rectangle and click the left button on the mouse.

Result: A new view is drawn showing the part enlarged.

Hit the **M** key to return to the **MAIN MENU**

STEP 5: FILLETS

From the **MAIN MENU** select the **ARC** function.

From the **ARC MENU** select the function **4. FILLET**

Move the cursor close to the middle of the right hand Line of the rectangle and click the left button on the mouse.

Now move the cursor close to the middle of the top Line.

Again, click the left button on the mouse.

Type **0.5** for the radius and hit the **Enter** key.

Result: A Fillet is drawn and the two Lines are trimmed automatically.

Next pick the two Lines forming the lower righthand corner of the rectangle in the same way but do not type in a value for the radius. Just hit the Enter key to accept the default value of 0.5.

Next draw the other two Filllets with radius **0.75** as follows:

Move the cursor close to the middle of the left Line and click the left button on the mouse.

Move the cursor close to the middle of the bottom Line and click the left button on the mouse to pick this Line.

Type 0.75 for the radius and hit the Enter key.

Hit the Esc key to return to the **ARC MENU**

STEP 6: DIMENSIONS (Actual Dimensions)

Press D to go directly to the **DIMENSIONS MENU.** (By using the quick keys, you can go directly from one menu to another).

Select function 1. **HORIZONTAL.**

The cursor should now be on the screen.

With the mouse, move the cursor near the bottom of the left Line of the rectangle and click the left button on the mouse to pick this Line.

Next, move the cursor close to the bottom of the right line of the rectangle and click the left button on your mouse to pick this Line.

Move the cursor to where you want the dimension and click the left button on the mouse.

Result: A horizontal dimension is drawn.

Hit the right button on your mouse to return to the **DIMENSION MENU.**

Select function **2.VERTICAL** and repeat the above steps to draw the Vertical Dimension. This time pick the right end of the top and bottom Lines.

Hit the right button on your mouse to return to the **DIMENSION MENU.**

STEP 7: RADIUS

Now dimension the Fillets as follows:

Select the function **RADIUS** in the **DIMENSION MENU.**

With the cursor pick one of the Fillets.

Next, place the Dimension where you want it.

Dimension the other Fillets in the same way.

If the drawing on your screen looks like the rectangle on the first page of this exercise, **WELL DONE!**

****

You can now delete everything as follows:

Hit E to go to the **EDIT MENU** (pressing the quick keys does not work when the cursor is on the screen. In that case, you need to hit the right button on the mouse or the Esc key to get out of the cursor mode.

Select function **1. DELETE**

Select **5. ALL** and answer **Yes.**

FASTER WAY TO DRAW A RECTANGLE

Press the O key to go to the **OTHER CURVES MENU.**

You will see the following menu:

OTHER CURVES MENU
1. **RECTANGLE**
2. **OFFSET**
3. **SPLINE**
4. **HOLE PATTERN**
5. **POCKET**
6. **CLEAN CIRC**
7. **TEXT PATH**
8. **GEAR**

Select function **1. RECTANGLE**

Select method **2. COORDINATES**

Type 1 for the X-coordinate of the lower left corner.

Type 1 for the Y-coordinate of the lower left corner.

Next, select method **3. INC X Y** (INCREMENT)

Type **5** for the **X** increment (length of the Rectangle)

Type **3** for the **Y** increment (height of the Rectangle).

Next do not type in a radius value but leave it at 0 and hit the Enter key. (If you type in a radius value each corner will get the same Fillet. Our Rectangle has different Fillets).

Result: A Rectangle is drawn of size 5x3.

Next draw the Fillets as shown in the previous exercise and also described below:

Hit **A** to go to the **ARC MENU.**
From the **ARC MENU** select the function **4. FILLET.**
The cursor is now on the screen.
Move the cursor close to the middle of the right Line.

Click the left button on the mouse.
Now move the cursor close to the middle of the top Line.
Again click the left button on the mouse.
Type 0.5 for the radius and hit the Enter key.
Do the same for the other corners using a radius of size 0.75 for the corners on the left.

As you can see this is a shorter way to draw a Rectangle.

Return to the **MAIN MENU** by hitting the right button on the mouse.

COPY/MOVE EXERCISE

The COPY/MOVE function is an extremely useful function. With this function you can COPY, MOVE, ROTATE, MIRROR, SCALE and DRAG Points, Lines , Arcs, Dimensions and Surfaces.

Before you start this exercise make sure that the screen is cleaned up as follows: You can exit **BOBCAD** and start from the beginning or you can delete everything if you do not want to exit.

If you have Elements on the screen and you want to delete them then proceed as follows:

Press **E** to go to the **EDIT MENU.**

Select function **DELETE.**

Select function **ALL** and answer Yes.

BEGIN:

Begin with a Zoom scale of 1. Do this as follows:

Press **Z** to go to the **Zoom Menu.**

Select function **NEW SCALE.**

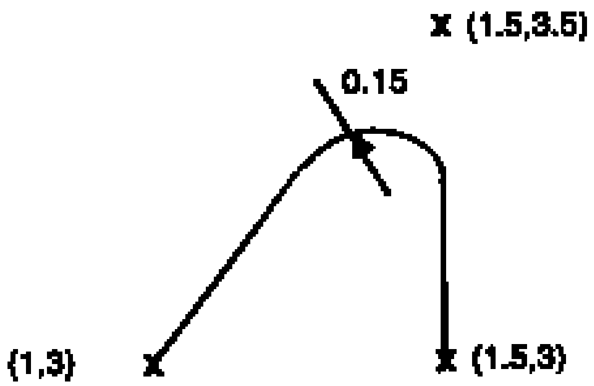
Type in 1 for the scale and hit Enter.

Hit the [key (as shown in the prompt line on the screen).

Type -1 for the X-ORIGIN then hit Enter

Type -1 for the Y-ORIGIN then hit Enter

Result: The X-Y axis is in the lower left corner of the screen.



Draw the tooth shown above as follows:

STEP 1:
Hit **P** to go to the **POINT MENU.**

Select function **2.COORDINATE** and enter 1 for the X-coordinate and 3 for the Y-coordinate.

Result: the left Point is drawn.

STEP 2:
Hit the **Esc** key or the right button on the mouse to return to the **POINT MENU.**

Select function 3.INC X-Y.

With the cursor pick the Point you have just drawn. It will turn red once picked.

Note: With BOBCAD the cursor doesn not need to be on top of the Point or Line or Arc that you want to pick. In fact, the cursor can be on the other side of the screen and it will find the nearest Entity.

Type in 0.5 for X.

Type in 0 for Y.

Result: A new Point is drawn to the right of the Point you picked.

Draw the third Point using the same function.

STEP 3:

Next: Hit the right button on the mouse to return to the **POINT MENU.**

Hit the L key to go to the **LINE MENU**

Draw the 2 Lines by using the **3.JOIN** function on the 3 Points as follows:

Select function **JOIN** from the **LINE MENU.**

Pick the lower right Point and then the top Point.

Then pick the lower left Point and the top Point.

Next hit the Esc key or the rigth button on your mouse to return to the **LINE MENU.**

STEP 4.

Press **A** to go to the **ARC MENU.**

Select **4.FILLET** and pick each line.

Draw the Fillet as shown with radius = **0.15.**

Next hit the **ESC** key or the right button on the mouse as needed to return to the **MAIN MENU.**

STEP 5.

Make 9 copies of the tooth as follows:

Select function COPY / MOVE from the MAIN MENU.

You will see the following menu:

*******COPY*******
1. **TRANSLATE**
2. **ROTATE**
3. **MIRROR**

*******MOVE*******
6. **TRANSLATE**
7. **ROTATE**
8. **MIRROR**

Select function **1.TRANSLATE** from the **COPY** Section.

Select function **BY POINTS** from **COPY: TRANSLATE. SELECT VECTOR OPTION.**

Next select function **1.SINGLE** and the cursor comes on the screen.

With the cursor pick the 2 Lines and the Arc. They will turn red when you select them.

STEP 6:

When you have picked all the Entities you need to hit the right button on the mouse.

Answer "NO" for More entities.

The cursor is now on the screen to establish the translation vector.

When it says "Indicate 'from' point" pick the left Point.

When it says "Indicate 'to' point" pick the right Point.

Type in 9 for the number of copies and then hit Enter.

Result: 10 teeth will appear as shown below.

EXERCISE IN MIRRORING

STEP 1:
Draw Line B as shown in the above drawing (Just use any function you like).

STEP 2:
After you have drawn Line B then:

From the **MAIN MENU** select function **COPY/MOVE.**

From the **COPY MENU** select function **MIRROR.**

The cursor is now on the screen.

Pick Line B as the mirroring Line.

Note: BOBCAD can Mirror in any Line. If you want to Mirror in the X or Y axis you must draw a Line that lies on the axis.

Pick all the teeth using either the **SINGLE, CHAIN** or **REGION** select method.

Result: All the teeth are drawn mirrored in Line B.

EXERCISE: ARC TOUCHING 3 ENTITIES

Clear the screen as follows: (as you learned earlier):

Hit M for the MAIN MENU.

Hit E for the EDIT MENU.

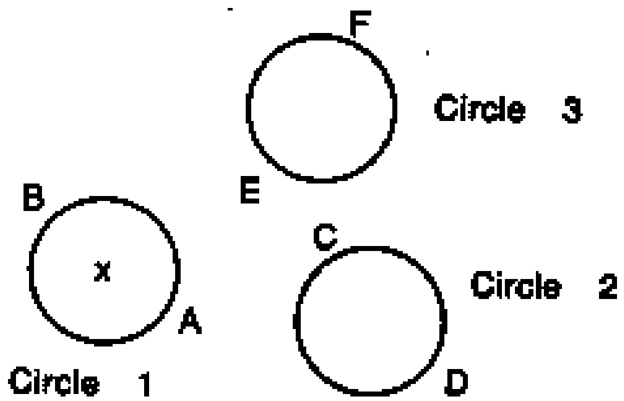
Select 1. DELETE

Select 5. ALL

Select 1. YES

Finally hit M for the MAIN MENU.

<u>In this Exercise we will draw an Arc that touches 3 Entities.</u>



First draw the above figure as follows:

<u>STEP 1: Draw Circle 1</u>
Press the A key to go to the ARC MENU.

Select function 1.SKTCH CNTR from the ARC MENU.

Type in 1 for the radius and hit the Enter key.

Do not type in a start angle but just hit the **Enter** key to accept the 0 value.

Hit the **Enter** key again to accept the 360 value.

The CIRCLE is now on the screen.

With the mouse move the CIRCLE to a position where you want the Circle to be and press the left button on the mouse to pick this position.

Result: Circle 1 is drawn.

STEP 2:

The cursor is still on the screen.

Do not type in any values but just hit the Enter key 3 times (or the left button on the mouse 3 times).

With the mouse move the Circle to a position where you want the center of Circle 2 to be and press the left button on the mouse.

Result: Circle 2 is drawn with the same radius.

Draw Circle 3 in the same way.

NOTE: MAKING A MISTAKE
If an Entity comes out differently from the way you wanted it, delete it as follows: Just press the 'DEL' key (it also has '.' on that key) to delete the last Entity you have drawn. The **NUM LOCK** key must be **ON** for this to work. Or: press the **E** key to go to the **EDIT MENU** and use function **DELETE LAST**.

STEP 3: Change the CURRENT LAYER

Press the right button twice on the mouse to go to the **MAIN MENU**.

Select function LAYER

You will see the folllowing menu:

LAYER MENU
1. **NEW**
2. **SHO ENTITY**
3. **MOD ENTITY**

4. **BLANK**
5. **UNBLANK**

ARC TOUCHING 3 ENTITIES

Next select function **NEW** from the **LAYER MENU.**

Type in 20 (or any other number up to 999) and hit the Enter key.

Result: *The number you have typed is shown in the lower left corner of the screen. From now on any new Entity you draw will be on this layer. You will see later on in this lesson why this is useful.*

STEP 4:
Press the A key to return to the **ARC MENU.**

Select function **5.3 ENTITIES** from the **ARC MENU.**

The cursor is now on the screen.

Move the cursor close to position 'A' (see the drawing at the beginning of this exercise).

Hit the left button on the mouse to pick Circle 1.

Next pick Circle 2 at position 'C'

Next pick Circle 3 at position 'E'.

Result: *An Arc is drawn that touches Circles 1, 2 and 3 at positions 'A', 'C' and 'E'.*

STEP 5:
The cursor is still on the screen.

Repeat the above process but this time pick Circle 1, 2 and 3 at positions 'B', 'D' and 'F' in this order.

Result: *a new Arc is drawn that touches the same Circles on the outside.*

Note: Remeber that in BOBCAD all Arcs are drawn couter-clockwise. In this function the first Entity that you pick will be the start of the new Arc and the last Entity that you pick will be the end of the new Arc.

STEP 6: BLANKING
Hit the right button on the mouse to return to the **ARC MENU.**

Press the **M** key to go to the **MAIN MENU.**

Select function **LAYER** from the **MAIN MENU.**

Select function **BLANK.**

Select function **LAYER #** and type in **20.**

Result: All Entities that you have drawn on layer 20 will now be blanked (they are not deleted, they are just made invisible)

Circles 1, 2, and 3 should still be on the screen!

If you want you can press the R key to **Repaint (Redraw)** the screen.

STEP 7: Pick the Circles at different positions.
Go back to the **ARC MENU** and again select Function **3 ENTITIES.**

Pick the Circles at different positions. For example, try these combinations: 'A', 'D', 'E', positions 'C' , 'F', 'B', positions 'B', 'C', 'E'.

STEP 8:
Again Blank layer 20 (as in STEP 6) so that only the 3 Circles remain on the screen.

Press the **P** key to go to the **POINT MENU.**

Select function **6. ARC CENTER** from the **POINT MENU.**

With the cursor pick Circle 1.

Result: A point is drawn at the center of Circle 1.

STEP 9:
Hit the right button on the mouse to return to the **POINT MENU.**

Press the **A** key to go to the **ARC MENU.**

Select function **5.3 ENTITIES** from the **ARC MENU.**

With the cursor pick the Point at the center of Circle 1.

Next with the cursor pick Circle 2 at C.

Next with the cursor pick Circle 3 at E.

Result: An Arc is drawn that touches the three Entities.

STEP 10: LIMITED SEARCH

First, with the cursor pick Circle 3 at F.

Next with the cursor pick the Point at the center of Circle 1 **BUT WATCH OUT!!**

When you move the cursor close to the Point to pick it , the cursor is just as close to the Point as to the inside Arc you have just drawn. Instead of pressing the left button on the mouse, press the P key on the keyboard.

Now **BOBCAD** knows to search for the nearest Point only without picking the Arc. (See the section called **QUICK KEYS** in this manual for other limited search keys).

Next pick Circle 2.

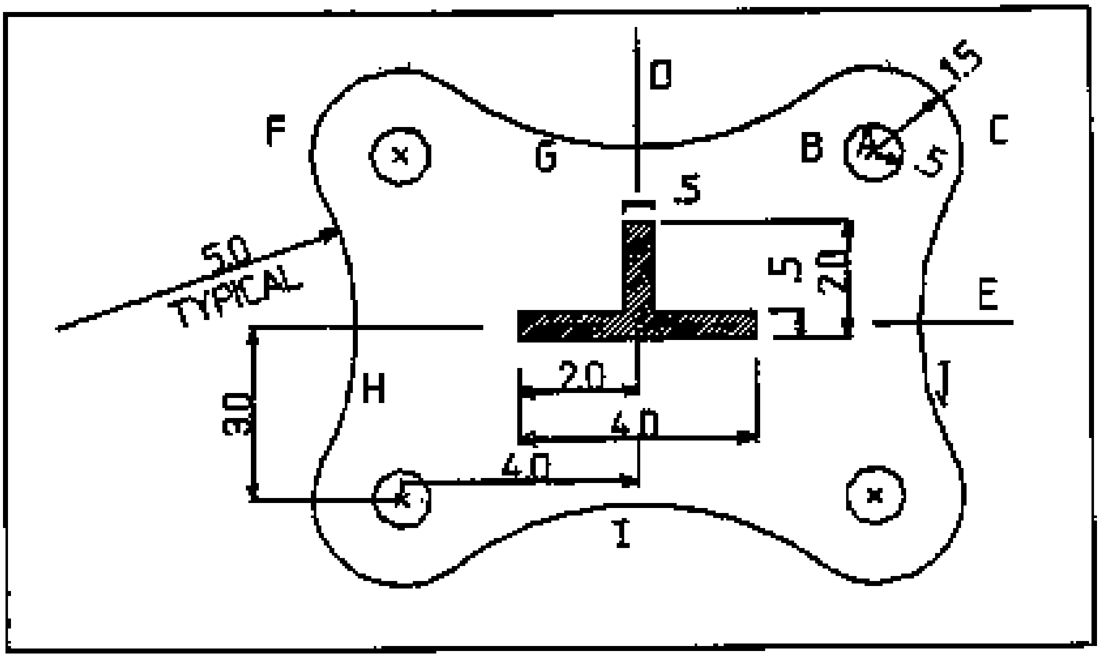
FINALLY: Hit the right button on the mouse to return to the **ARC MENU.** Next hit the **M** key to go to the **MAIN MENU.** Select function **LAYER** and UNBLANK layer 20.

SUMMARY: The function **3 ENTITIES** and **FILLET** can draw Arcs touching any combination of Points, Lines, and Arcs.

ADDITIONAL EXERCISES: You can do some more by trying out the function **3 ENTITIES** on 3 Points or 3 Lines.

DRAWING A PART
(INTERMEDIATE EXERCISE)

This part can be drawn quickly because it is symmetrical. After you have drawn it once, delete it and draw it again to get more experience. You will find you draw it in half the time the second time around.



Go to the **POINT MENU** and select **2.COORDINATE.**

1. Draw Point 'A' at X = 4, Y = 3.

2. Draw Circle 'B' as follows:

In the **ARC MENU** use function **POINT CENTER.**

Next enter radius = 0.5.

Next do not type in the start and end angle but just hit the Enter key.

With the cursor pick Point 'A'

3. Continue with the same function to draw Arc 'C' as follows: Enter radius = 1.5. Enter start angle = -10, end angle = 110. Next pick Point 'A' again.

Note: the start and end angles that we typed in are just a guess. Arc 'C' will get trimmed later on in this exercise.

MIRROR LINES
BOBCAD lets you mirror in any Line at any angle and position. This does require that a Line is there. Draw Lines 'D' and 'E' now. For example draw Line 'D' on the Y-AXIS with coordinates **XS = 0, YS = 0, XE = 0, YE = 3.** *(Note the Line is drawn on top of the Y-AXIS. You can see the Line better if the ACTIVE COLOR is different from the MENU COLOR. Another option is to turn the X-Y AXIS off in the CUSTOM MENU).*

Draw a horizontal Line (Line 'E') on the X-AXIS. For example, with coordinates XS = 0, YS = 0, XE = 5, YE = 0.

MIRRORING
In the **MAIN MENU** select the function **COPY/MOVE.**

Next, select **COPY 3. MIRROR** and pick Line 'D' as mirroring Line.

Next, select **SINGLE** from the menu and pick Circles 'B' and 'C'.

Next, hit the right button on the mouse. Answer **NO** to selecting more Entities and you will get the mirrored Circles.

FILLETS
To draw Fillet 'G' use the **FILLET** function in the **ARC MENU** as follows:

After you select function **Fillet**, pick Circle 'F' at the top approximately where Fillet 'G' starts from. Next pick Circle 'C' approximately where Fillet 'G' would end. (Remember that Arcs, Circles and Fillets always are drawn counter-clockwise). Next enter radius = **5.**

As you can see Circles 'F' and 'C' got trimmed automatically.

MIRRORRING AGAIN
Use the **MIRROR FUNCTION** again to Mirror everything in Line 'E'. (You can use **REGION** to pick the Entities).

After you have mirrored do a Zoom as needed.

MORE FILLETS
Draw Fillets 'H' and 'J' in the same way as above. Remember to pick as the first Circle where the Fillet will start , and as the second Circle where the Fillet will end. This time you do not have to enter the radius because

when the system asked you for the Radius size, it told you that the default was 5. This means you just have to hit the Enter key or the left button on the mouse to accept that value.

<u>CENTER ISLAND</u>

Draw the Center Island as follows: Select **8.CONTINUOUS** in the **LINE MENU.**

Select **2.COORDINATE.**

Type in the start coordinates of the string: X = -2 , Y = -.25

Select function 3.INC X , Y

TYPE IN: INC X = 4
INC Y = 0

NEXT TYPE IN: INC X = 0
INC Y = 0.5

NEXT TYPE IN: INC X = -1.75
INC Y = 0

NEXT TYPE IN: INC X = 0
INC Y = 1.55

NEXT TYPE IN: INC X = -0.5
INC Y = 0

NEXT TYPE IN: INC X = 0
INC Y = -1.55

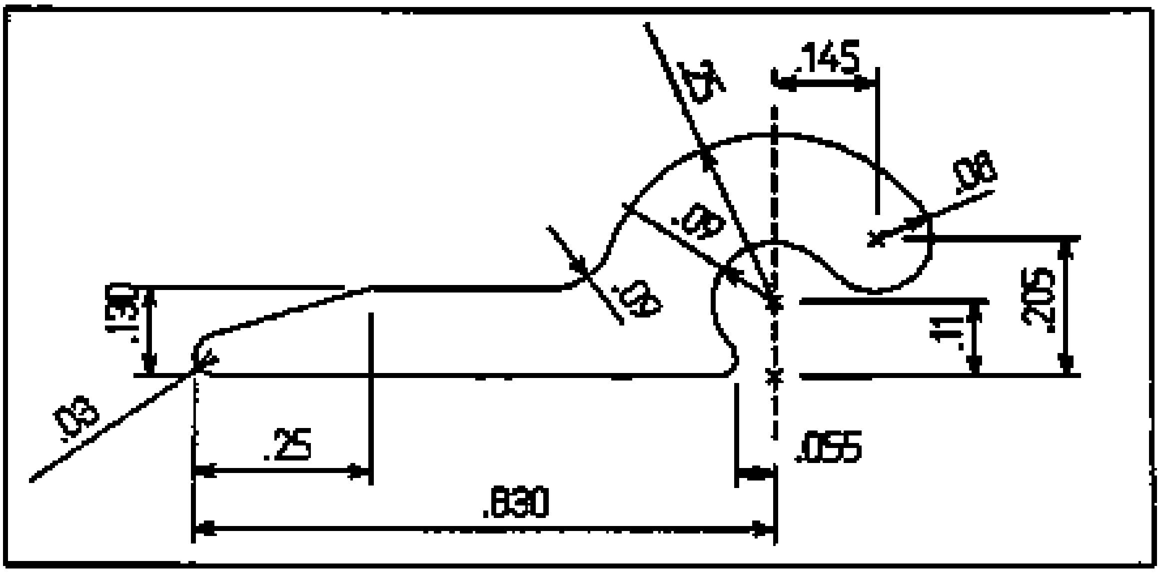
NEXT TYPE IN: INC X = -1.75
INC Y = 0

Next Hit the Esc key.
Next select function JOIN and with the mouse pick the beginning of the Continuous Line.

--END OF EXERCISE--

ADVANCED EXERCISE

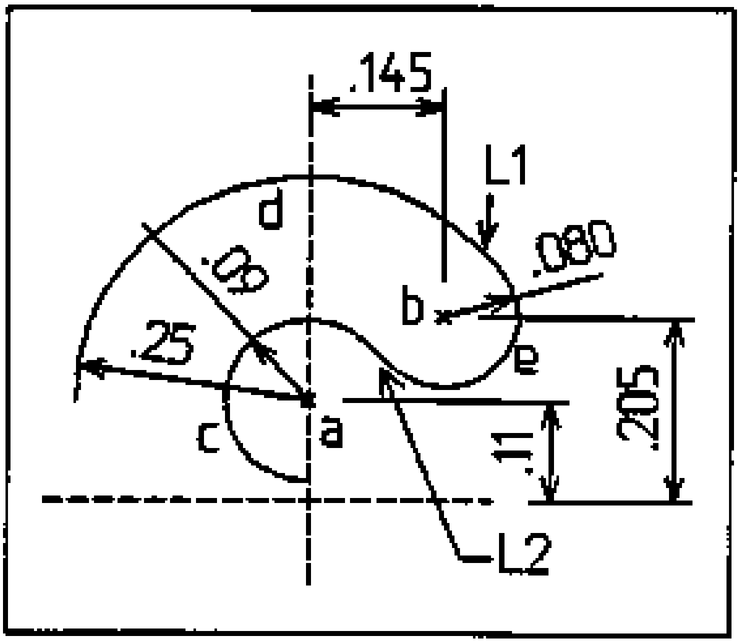
This is a more advanced part and requires that you are familiar with the earlier exercises.

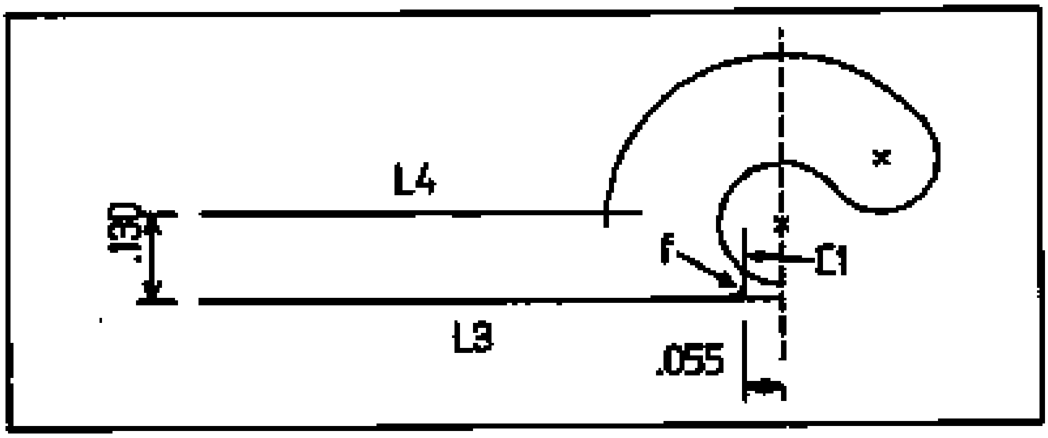


Start a new drawing.

1. Use the POINT COORDINATE function to draw Point 'a' at X = 0, Y = 0.11 and Point 'B' at X = 0.145, Y = 0.205
2. Use the ZOOM: NEW SCALE function: enter 0.35 for the new scale. Pick the origin as the new center.
3. Use function ARC:POINT CENTER to draw Arc 'c' around Point 'a' with radius = 0.09, start angle = 60 and end angle = 270. Draw Arc 'd' around Point 'a' with radius = 0.25, start angle = 70 and end angle = 180. (These are aproximate angles).

   Draw Arc 'e' around Point 'b' with radius = 0.08, start angle = 180 and end angle = 30.
4. Use ZOOM: VIEW ALL to enlarge the 3 Arcs.
5. Use the LINE TANGENT function to draw Lines 'L1' and 'L2'.





6. Use the TRIM: ONE ENTITY function to trim Arc 'c' to Line 'L2'.
   Trim Arc 'E' to Line 'L2'
   Trim Arc 'D' to Line 'L1'.
   Trim Arc 'E' to Line 'L1'

   Note1: If your trim goes wrong then hit the BACK SPACE key on your keyboard to restore the previous Entities. This only works while you are still in the trim function.

   Note2: IMPORTANT. It is important that when you pick the Entity that you want to trim that you place the cursor close to the intersection of the two Entities.

7. Draw Line 'L3' on the X-axis using the LINE: COORDINATE function (use approximations for the X values).

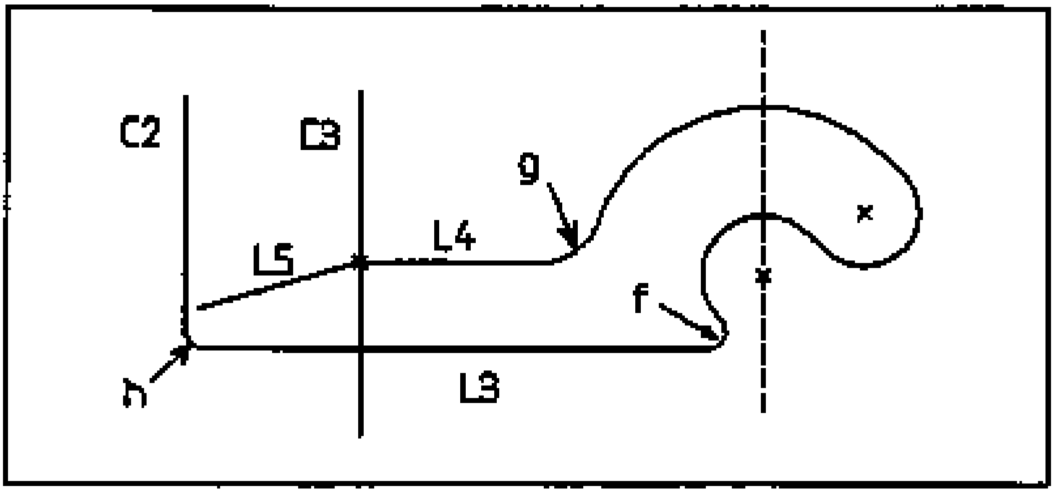
8. Draw a vertical construction Line 'C1' with coordinates XS = -.055, YS = 0, XE = -.066, YE = 0.1

9. Use function ARC: 3 ENTITIES to draw Arc 'f' (The exercise ARC: TOUCHING 3 ENTITIES teaches you how to use this function)

10. Trim Line 'L3' to Arc 'F'
    Trim Arc 'C' to Arc 'F'
    Delete the Vertical Line 'C1'.

11. Twice do a ZOOM:REDUCE.

12. Draw Line 'L4' using the LINE :SKETCH function with Y = .13



13. Draw Fillet 'G' with radius = .09

14. Draw Arc 'H' as follows:
Use the LINE SKETCH function to draw a Vertical construction Line 'C2' with X value = -.83. Make sure the Line starts below Line 'L3' and ends above Line 'L4'.

15. Use function LINE:PARALLEL to draw a Line 'C3' at 0.25" to the right of construction Line 'C2'.

16. Use the TRIM: ONE ENTITY function to trim Line 'L4' to construction Line 'C3'.
Use function POINT: END to draw a Point at the left end of 'L4'

17. Use the ARC:FILLET function to draw Arc 'H' with radius = 0.03 between 'L3' and 'C2'.

18. Use the LINE TANGENT function to draw 'L5' between the last Point that you have drawn and Arc 'H'.

19. Delete construction Line 'C2' and 'C3'

20. Use the TRIM:ONE ENTITY function to trim Arc 'H' to Line 'L5'

-----------------------------END OF EXERCISE-----------------------------

SPHERE

3-D EXERCISE

1. Draw a Point at X = 0 , Y = 0

2. Draw a Circle at X = 0, Y = 0, Radius = 3, Start angle = 0 , End angle = 359.

3. Select function THREE - D from the UTILITIES MENU.
Next: select function **TURN 3 - D ON.**

---

4. Next: hit the Z key to go to the ZOOM MENU.
Next: select function **3 - D VIEWS.**
Next: select function **FRONT X - Z.**
Next: hit the right button on your mouse to return to the ZOOM MENU.
Next: select function **REDUCE** in the ZOOM MENU once.

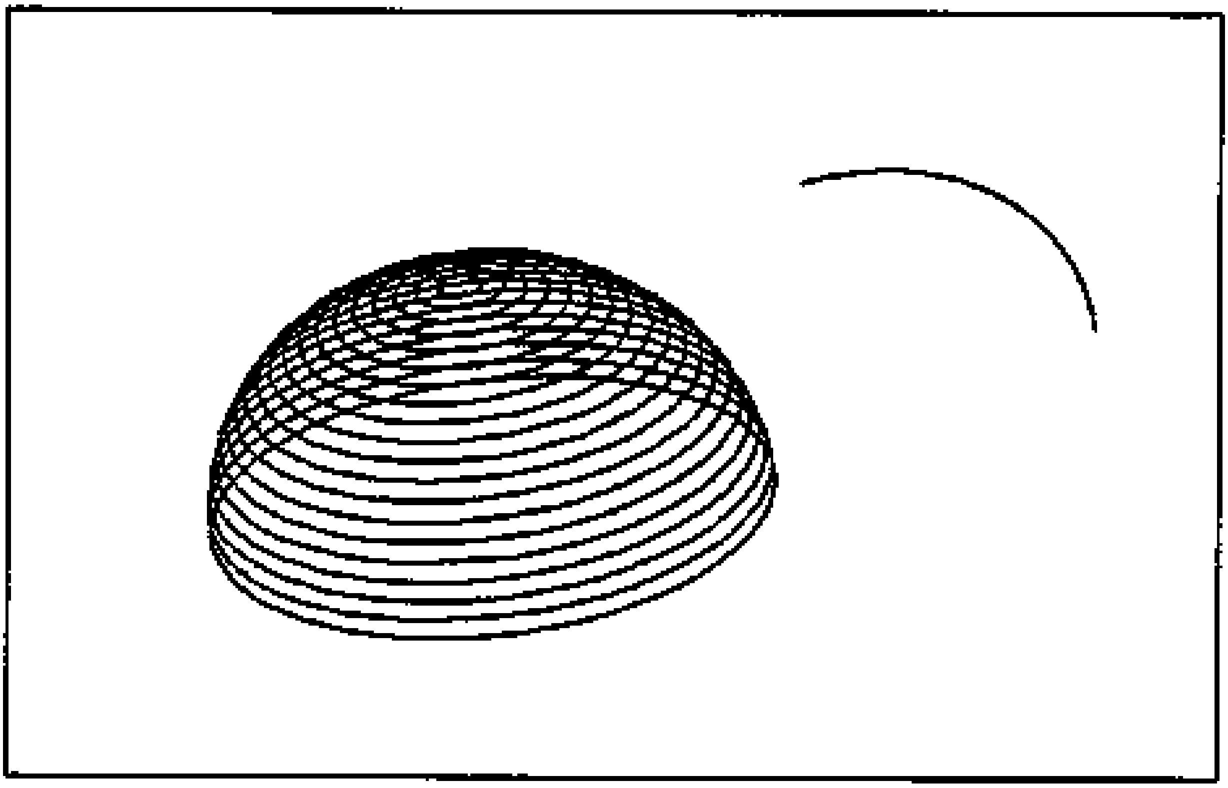
---

5. In the ARC MENU select function **COOR CNTR.**
Draw an Arc at X = 4, Y = 0, Z = 0, Radius = 3, Start Angle = 0, End angle = 90.
(Note the Arc is drawn in the plane of the screen, which is the X-Z plane)

6. Hit the Z key to go to the ZOOM MENU.
Next: select function **3-D VIEWS.**
Next: select function **DEFAULT.**

7. Hit the U key to go to the UTILITIES MENU.
Next: select function **THREE-D**
Next: select function **INTERPLT.**
Set the number of divisions to 20.
Select SINGLE and pick the Arc that you have drawn last.

8. Next: select function **SURFACE** from the 3-D **MAIN MENU**.
Next: select function **FOLLOW CURVE**.
Pick the full Circle as the contour to be offset.
Pick the inside of the Circle as the side of the tool.
Next: hit the right button on the mouse when it asks INDICATE NEXT / LAST ENTITY as there is only one entity.
Answer YES to the question SELECTION OK ?
Next: pick the right bottom of the 90 degree Arc (which is now actually a chain of 20 Lines) and indicate direction up along the Arc.
Hit the] key (next to the Enter key) to go to the end of the curve untill it is completely red.
Result: A sphere is drawn



<u>DISH</u>

<u>3-D EXERCISE</u>

0. Start a new drawing.

1. Draw a Circle at X = 0, Y = 0, Radius = 3, Start angle = 0, End angle = 359.

2. Select function **THREE - D** from the **UTILITIES MENU.**
Next: select function **TURN 3 - D ON.**

---

3. Next hit the **Z** key to go to the **ZOOM MENU.**
Next: select function **3 - D VIEWS.**
Next: select function **FRONT X - Z.**
Next: hit the right button on your mouse to return to the **ZOOM MENU.**
Next: select function **VIEW ALL** in the **ZOOM MENU.**
Next: select function **REDUCE** in the **ZOOM MENU** twice.

---

4. In the **ARC MENU** select function **COOR CNTR**
Draw an Arc at X = 6, Y = 0, Z = 4, Radius = 5, Start Angle = 270, End angle = 310.
(Note the Arc is drawn in the plane of the screen which is the X-Z plane)

5. Draw a Line with coordinates XS = 4, YS = 0, ZS = 0, XE = 8, YE = 0, ZE = 0

6. Hit the **T** key on your keyboard to go to the **TRIM MENU.**
Select function **TWO ENTITY** and trim the Arc to the Line.

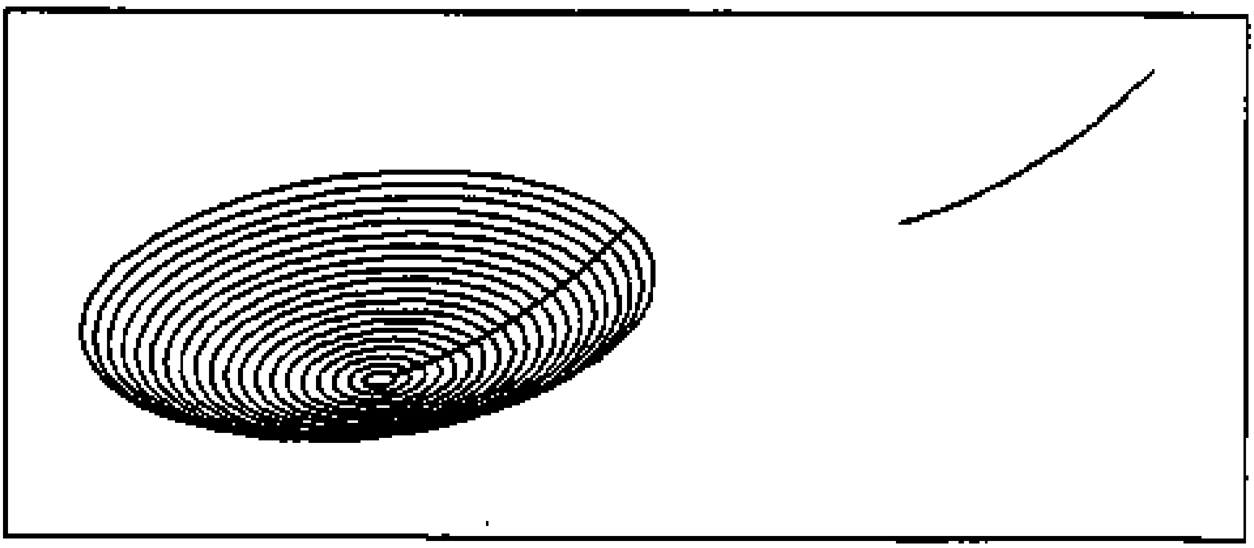
7. Hit the **Z** key to go to the **ZOOM MENU.**
Next: select function 3-D VIEWS.

Next: select function **DEFAULT.**

8. Next: use the **DELETE MENU** to delete the Line.

9. Hit the U key to go to the **UTILITIES MENU.**
Next: select function **THREE-D**
Next: select function **INTERPLT.**
Set the number of divisions to 20.
Select SINGLE and pick the Arc that you have drawn last.
Result: the Arc is now broken into 20 Line segments.

10. Next: select function **SURFACE** from the 3-D MAIN MENU.
Next: select function **FOLLOW CURVE.**
Pick the full Circle as the contour to be offset.
Pick the inside of the Circle as the side of the tool.
Next: hit the right button on the mouse when it asks INDICATE
NEXT / LAST ENTITY as there is only one entity.
Answer **YES** to the question SELECTION OK ?
Next pick the right top of the Arc (which is now actually a chain
of 20 Lines) and indicate direction down along the Arc.
Hit the] key (next to the Enter key) to go to the end of the curve
untill it is completely red.
Result: A dish is drawn



CAM EXERCISES

Section 3

3.1) CAM EXERCISE #1 (with editing)

3.2) CAM EXERCISE #2

3.3) ROUGHING A PART

3.4) 3-D PART: RULED SURFACE

3.5) 3-D PART: P - TRAP

3.6) 3-D PART: CROSS-SECTION

3.7) EXERCISE BY GARY SVENSON OF SVENSON MACHINE. NC PROGRAM INCLUDED.

3.8) EXERCISE BY GARY SVENSON FOR DYNAPATH DELTA 10 CONTROL. NC OUTPUT FILE AND MILLCFG ARE INCLUDED.

CAM EXERCISE #1

This is a simple 2.5 Axis exercise which shows you how to draw an outline and then machine it.

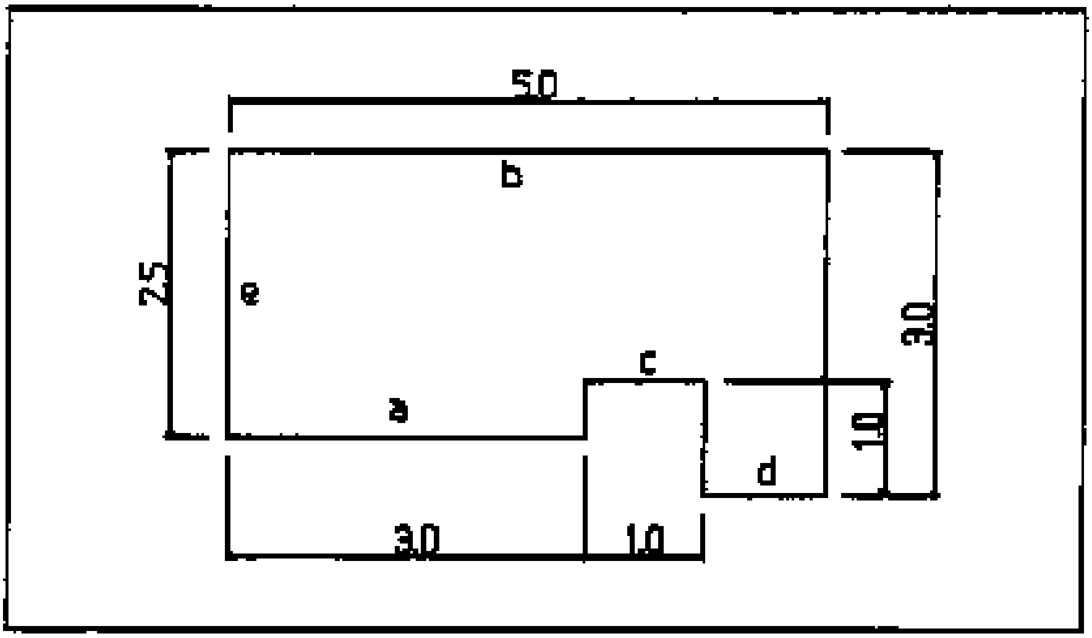
From DOS type : CD \BOBCAD (and hit the Enter key) to go to directory BOBCAD.
Next type BOBCAD to start BOBCAD.

Now you will see the START OPTIONS MENU on the screen.
Hit the number 1 on your keyboard to select option 1.START A NEW DRAWING.

Now you will see the X-Y Axis on your screen.
Type in a name for the drawing for example: TEST and hit the Enter key.

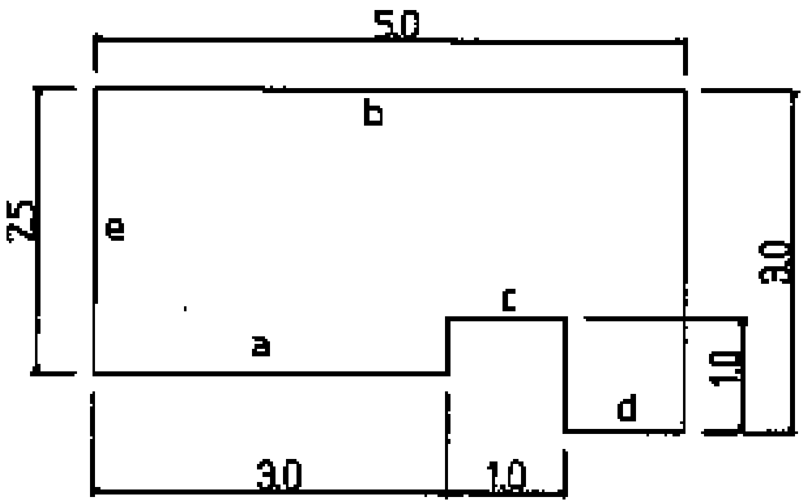
The MAIN MENU is now on the screen.

Draw line A (see drawing below) as follows:



With the mouse move the hi-light in the menu up or down untill you are on the LINE function. Then click the left button on the mouse.
You are now in the LINE MENU.
Next: With the mouse point at function 2.COORDINATE in the LINE MENU (and click the left button on the mouse again).

CAM EXERCISE #1



Next type in the following coordinates:

Type in the value 1 for the X START COORDINATE (and hit the Enter key)

Type in the value 1 for the Y START CORRDINATE.

Type in the value 4 for the X END COORDINATE.

Type in the value 1 for the Y END COORDINATE.

Result: Line 'A' is drawn on the screen.

Next draw line 'B' using the following coordinates:

X-START = 1
Y-START = 3.5
X-END = 6
Y-END = 3.5

Before we continue lets ZOOM IN as follows:

Hit the 'Z' key on your keyboard to go to the ZOOM MENU.

Next select function WINDOW from the ZOOM MENU.

With the mouse move the cursor to about X = 0 and Y = 0 and then click the left button on the mouse.

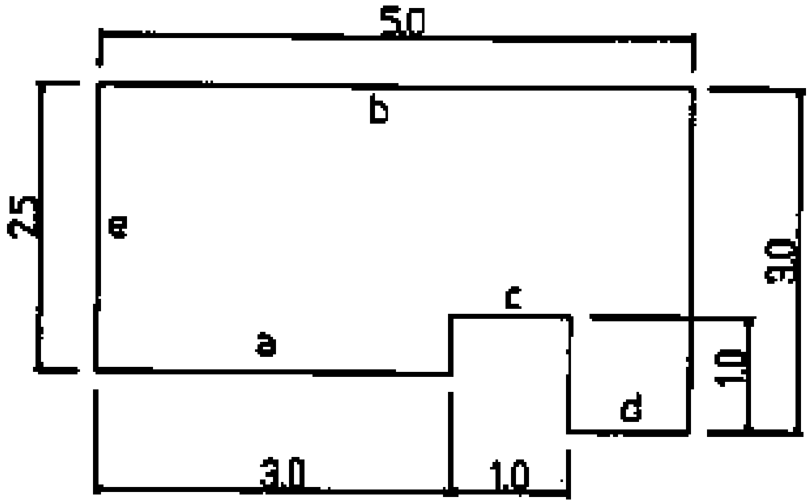
Next with the mouse move the cursor so that you get a box around the two lines on the screen leaving a lot of space and then click the left button on the mouse again.

Result: The part appears enlarged.

Note: If the ZOOM scale is not the way you want it then use the other functions in the ZOOM MENU untill it is the way you want it.

Next hit the right button on the mouse to exit the ZOOM MENU and to return to the LINE MENU.

CAM EXERCISE #1



Next draw line 'C' using the following coordinates:

X-START	=	4
Y-START	=	1.5
X-END	=	5
Y-END	=	1.5

Next draw line 'D' using the following coordinates:

X-START	=	5
Y-START	=	0.5
X-END	=	6
Y-END	=	0.5

Next hit the right button on the mouse to exit the LINE COORDINATE MENU and return to the LINE MENU.

From the LINE MENU select function JOIN.
Next with the mouse move the cursor to the left end of line A and hit the left button on the mouse to pick that end.
Next in the same way with the mouse and the cursor pick the left end of line B.
Result: line E is drawn.

Next with the cursor pick the right end of line A.
Next pick the left end of line C.

Continue to join the ends of the other lines untill you have the final shape as shown in the above drawing.

When you are done hit the right button on your mouse or the Esc key on your keyboard.

If your part looks like that in the above drawing, **WELL DONE !** It is now ready for the CAM MENU.

MACHINING THIS PART USING THE CAM PORTION OF BOBCAD.

Hit the 'N' key to go to the CAM portion of BOBCAD
Next select function NEW FILE to start a new NC program.
Type in a name for the NC output file for example: TEST and hit the Enter key.
You are now in the MILLING MAIN MENU.

Select function POINT MOVE from the MILLING MAIN MENU.
Next with the cursor pick the lower left corner of the part.
Result: A rapid move is made to the part and the code is shown on the bottom of your screen as G00 X1.0000 Y1.0000 Z.1000

Next: Select function TOOL UP/DN from the MILLING MAIN MENU.
Note that the default cutting depth is shown in function 5.Z=-1.2500.
Next: Select function TOOL DOWN.
Next: Because the feed rate is still at rapid you will get a warning to select the FEED MODE.
Result: The tool plunges down to the cutting depth and the code is shown as G01 Z-1.2500.

Next: Select function SINGLE from the MILLING MAIN MENU.
Next: With the cursor pick line 'A' which will turn red.
Next: Move the mouse to the right end of line 'A' so that the arrow points to the right and click the left button on the mouse
Result: line A gets cut and the code is added to the bottom of the screen as (G01) X4.0000

Next: select function SINGLE in the MILLING MAIN MENU again.
Result: BOBCAD automatically finds the next move.

You can use SINGLE step or you can use AUTO to cut the rest of the part.
Lets use AUTO to cut all the way around the part as follows:
Select function AUTO from the MILLING MAIN MENU.
Next: with the cursor pick line E as the last line to be cut.
Result: the part is cut up to line E.

If you were to Quit right now out of BOBCAD this NC program would get saved under the name TEST.TAP. Then run GEOTALK to send it to your CNC machine. But don't quit yet.....

Note: The NC can be configured by running MILLCFG.

.....continued

EDITING THE PART

Unlike other CAD-CAM systems, with BOBCAD you can EDIT the NC program while you create it. Do this as follows:

While in the MILLING MAIN MENU hit the **F2** key on your keyboard.
Next from the EDITOR MENU select function **EDIT BLOCK.**
Result: A text cursor appears on the bottom line of code and an arrow points to the Line that this code belongs to.

Move the cursor up with the mouse. The red arrow will move to show you were you are on the part.

For example move the text cursor all the way up untill it can go no further. Now move it down so that it is on the first G01 move (where the tool plunges into the part)
Next: with the mouse bring the text cursor to the end of that block .
Next: click the left button on the mouse so the bright text cursor is there.
Next type in a feed rate, for example: F20.0

Hit the **Esc** key or the right button on the mouse to exit the Editor.
Hit the right button on the mouse again to return to the MILLING MAIN MENU.

Use the TOOL UP/DN function to bring the tool back up again.

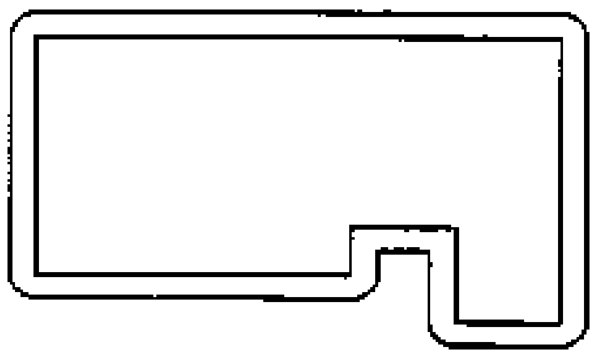
This was a simple straight forward cutting of a part, no cutter compensation etc. You can now quit out of BOBCAD and the NC program will be saved under the name TEST.TAP.

CUTTER COMPENSATION

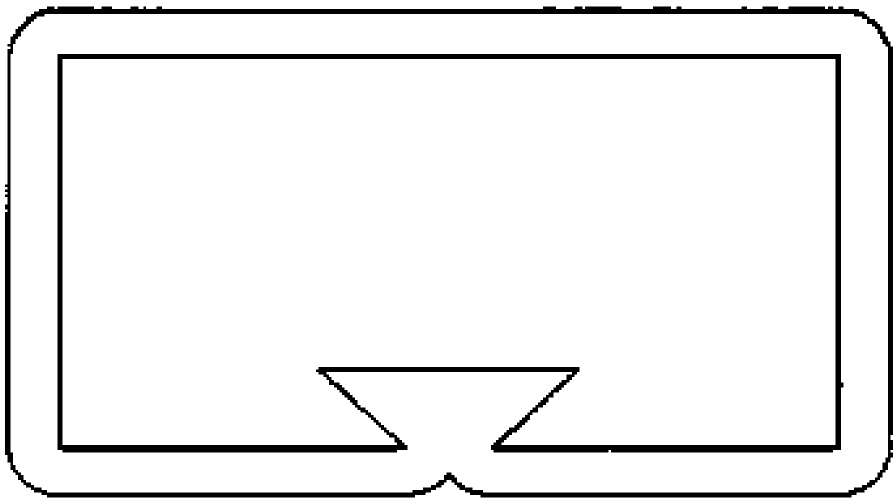
If your CNC machine accepts G41 and G42 codes then you would be drawing the outline of the final part and use G41 and G42 codes to tell the CNC machine to offset the tool.
If your CNC machine does not accept offset commands or you prefer to program tool center then in BOBCAD there are several functions that let you draw tool center very rapidly as shown in the following example.

OFFSET CURVES:
We will now use the 2-D shape that you have drawn in the first exercise.
We are going to machine this part on the outside.
Hit the 'O' key to go to the OTHER CURVES MENU.
Next select function OFFSET from the OTHER CURVES MENU.
Next select function CHAIN
Next enter OFFSET DISTANCE = 0.25 (Half the tool diameter)

Next with the cursor pick line 'A' so it turns red.
Next move the mouse under Line 'A' and near the right end of line 'A' so that the tool will be on the outside of the part and direction points to the right (then click the left button of the mouse).
Next hit the ']' key (square bracket next to the Enter key on your keyboard) to go to the end of the shape.
Result: the whole part turns red.
Answer YES to the question SELECTION OK ?
Result: an Offset curve is drawn on the outside of the part which you can now use as the tool center path. (see drawing below)



Note: As you can see BOBCAD OFFSET CURVES automatically generate an Arc around each sharp corner for smoother machining. Function OFFSET CURVES also checks for tool interference as shown in the part below with the tool not going into the dovetail because the diameter is too large.



CAM EXERCISE #2

This is another simple 2.5 - axis part.

Please note the following: This exercise will go through many of the functions of **BOBCAD**. People who have programmed N.C. machines before or are familiar with CAD or CAM systems will find this an easy exercise.

****

If you are having difficulties doing this part, work through the exercise once and then do it again and once more. You will find you get faster and faster.

****

Start **BOBCAD** as follows:

Note: Before you start please note: Every time you turn on your computer you must initialize your mouse as described in the manual that came with your mouse (Generally this is done by typing 'MOUSE' but your computer may be set up to do this automatically)

Type: **CD \BOBCAD** (and hit the Enter key).

This will bring you into the Directory where **BOBCAD** sits.

Type: **BOBCAD.**

Now you will see the **START OPTIONS MENU** on the screen.

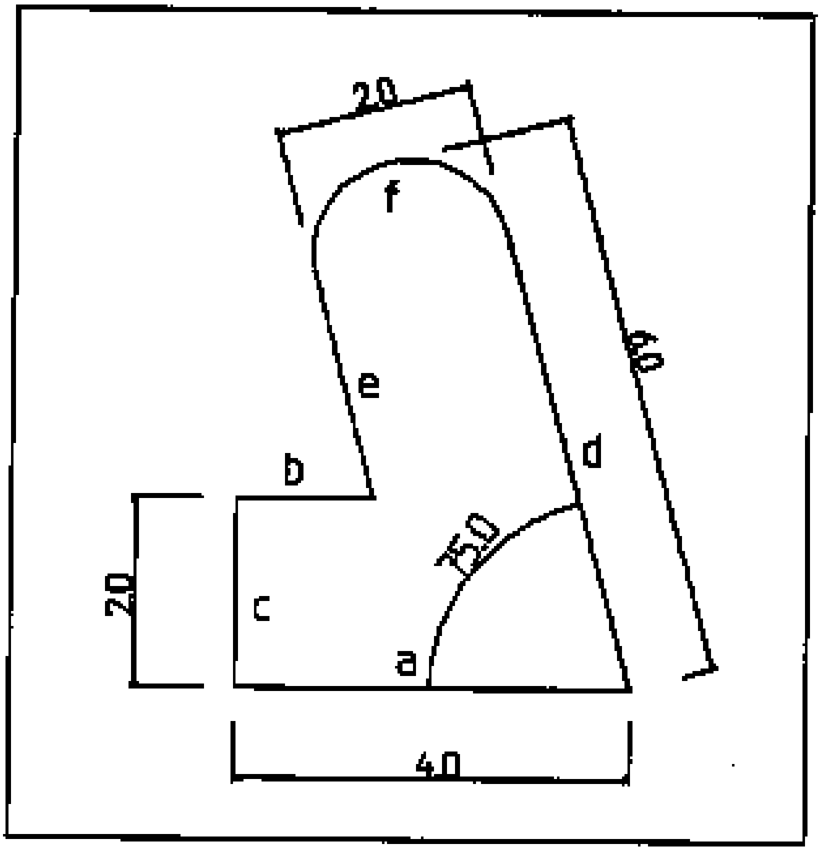
Hit the number 1 to select option '**1.START A NEW DRAWING**'.

Now you will see the X-Y Axis on the screen.

Type in a name for this drawing for example **EXERCISE** and hit the **Enter** key.

The **MAIN MENU** is now on the screen.

We will draw line A (see drawing) as follows:



With the mouse move the hi-light in the menu up or down until you are on the LINE function. Then click the left button on the mouse.
Result: the LINE MENU is on the screen.

(If you made a mistake and selected the wrong function hit the Esc key [left top keyboard] to return to the **MAIN MENU**. Then start again from there).

With the mouse, point at function **2.COORDINATE** in the **LINE MENU** (and click the left button on the mouse).

Next:
Type in the value **1** for the X START COORDINATE (and hit the Enter key).

Type in the value 1 for the Y START COORDINATE (and hit the Enter key).

Type in 5 for the X END COORDINATE.

Type in 1 for the Y END COORDINATE.
Result: Line 'A' is drawn on the screen.

CAM EXERCISE #2

Hit the Esc key (top left of keyboard) to return to the **LINE MENU**.

Now draw line'B' on the drawing as follows:

From the **LINE MENU** select function **5.PARALLEL.**

Result: the cursor is on the screen.

With the mouse move the cursor to Line 'A' and click the left button.

Result: Line 'A' now has a red color to show you that you have picked it.

Next move the cursor above line A and click the left button on the mouse again.

Next type in the distance 2 (and hit the Enter key).

Result: Line 'B' is drawn parallel to line 'A' (2" above it).

Now hit the Esc key or the right button on the mouse to return to the **LINE MENU**.

Draw the Line 'C' as follows:

Select function **3. JOIN** from the **LINE MENU**.

The cursor is back on the screen.

With the mouse pick the left ends of Line A and B to join them.

Result: Line 'C' is drawn.

Next hit the right button on the mouse (same as Esc key). To return to the **LINE MENU**.

Now draw Line 'D' using the angle function as follows:

Select function **6.ANGLE** from the **LINE MENU.**

Next with the mouse pick the right hand end of line 'A'.

Next type in the ANGLE 105 (and hit the Enter key). Type in length 6.

Result: Line 'D' is drawn.

Hit the **Esc** key or the right button on the mouse to return to the **LINE MENU.**

Next draw Line 'E' (Using the **5. PARALLEL** function again) as follows:

Select function **5. PARALLEL** from the **LINE MENU.**

With the cursor pick Line 'D' (which will turn red).

Next move the cursor to the left of Line 'D' and click the left button to indicate that side.

Next accept the distance of 2 by hitting the **Enter** key or the left button on the mouse (the program remembered the value of 2 from the last time).

Result: Line E is drawn.

Next hit the Esc key (or the right button on your mouse) to return to the **LINE MENU.**

Now join the top ends of Lines D and E using the **3. JOIN** function from the **LINE MENU** the same way as you did before when we drew Line C. (Look at the instructions on the bottom of your screen to remind you what to do).

(Line F is not shown on the drawing but we will use it for drawing Arc 'F').

After you have drawn the line using 3. JOIN function hit the **Esc** key to return to the **LINE MENU.**

CAM EXERCISE #2

Quick Keys

The purpose of the Quick Keys is to go to a menu instantly. The Quick Keys are shown as colored letters in the Menus.

Hit the **A** key to go to the **ARC MENU** instantly.

Now draw Arc F using function **5.3 ENTITIES** by selecting this function from the **ARC MENU**.

With the cursor pick Line D, next pick Line F, next pick Line E.

Result: Arc F is drawn.

Hit the right button on the mouse to return to the **ARC MENU**.

Trimming

Now we will trim Lines E and B to each other as follows:

Hit Quick Key **T** to go to the **TRIM MENU**.

Select function 3. **TWO ENTITY** from the **TRIM MENU**.

With the cursor pick Line E and Line B (make sure you pick them on that part of the line you want to keep. If the result is wrong because you picked the wrong part, then hit the **Backspace** key to get the original lines back again or use the trim function to trim them back).

Next hit the **Esc** key or the right button on the mouse to return to the **TRIM MENU** (The right button works the same as the **Esc** key).

Trim lines E and D as follows:

Select Function 2. **ONE ENTITY** from the **TRIM MENU**.

With the cursor pick Line E below Arc F.

Next pick Arc F as the border to trim to.

Result: Line E is trimmed to Arc F.

With the cursor pick Line D.

Next pick Arc F as the border.

Result:* *Line D is trimmed to Arc F.

Hit the right button on the mouse to return to the **TRIM MENU.**

Hit the Quick Key **E** to go to the **EDIT MENU.**

Select function **1. DELETE** from the **EDIT MENU.**

Select function **1. SINGLE** from the **DELETE MENU.**

With the cursor delete Line F as follows: Bring the cursor close to Line F and hit the **L** key on the keyboard. (By hitting the **L** key you make sure you picked the line and not the arc. However, if you are sure the cursor is closer to the Line than the Arc you can use the left button on the mouse).

Next hit the button on the mouse to return to the menu.

Result:* *The part is complete.

Hit the **R** key to Repaint the picture. This is not really necessary but it makes the picture look clean.

MACHINING THIS PART USING THE CAM PORTION OF BOBCAD

Hit the N key to go to the **CAM: NC MENUS.**

Select function 1. **NEW FILE.**

Type in any name for the NC output file (for example: TEST).

Result: The MILLING MAIN MENU is now on the screen.

With the mouse move the hi-light to function **'POINT MOVE'** in the menu and click the left button on the mouse.

With the mouse move the cursor to the lower left corner of the part and click the left button.

Result: A rapid move is drawn and the code is shown on the bottom of your screen as G00X1.0000 Y1.0000.

Next move the hi-light to function TOOL UP/DN and click it. You are now in the TOOL UP/DOWN menu. Now move the hi-light to function TOOL DOWN and click it. (Select FEED MODE).

Result: The tool plunges down and the code is shown at the bottom of your screen as G01Z-1.2500.

Next move the hi-light to function **'SINGLE'** (and click it).

With the cursor pick Line A it turns red.

Next move the mouse to the right so the arrow points to the right and then click the left button.

Result: The line gets cut and the code is added to the bottom of the screen as (G01)X5.0000.

Next click function 'SINGLE' in the MENU again.

Result: BOBCAD automatically finds the next move. Line D gets cut and the code is added at the bottom of the screen.

You can use 'SINGLE' step or you can use 'AUTO' to cut the rest of the part. Let's use 'AUTO' to cut all the way around the parts as follows:

With the hi-light go to function 'AUTO' and click it.

Now pick Line C as the last line.

Result: The part is cut.

EDITING THE NC PROGRAM

Unlike other CAD-CAM software, with **BOBCAD** you can edit the NC program while you create it. Do this as follows:

Hit the 'F2' key on your keyboard while you are in the **MILLING MAIN MENU**. Next select function EDIT BLOK.

A text cursor appears on the bottom line of code and an arrow points to the Line or Arc that this code belongs to.

Move the text cursor up with the mouse. The red arrow will move to show you where you are on the part.

For example: move the text cursor to the end of block G01X5.0000 (the block that is below the Z-1.2500 move). Next click the left button on the mouse so the bright text cursor is there. Next type in a feed rate for example: F20.0.

Hit the Esc key or the right button on your mouse to exit the Editor.

Now try out the other functions in the EDITOR MENU. The TOOL CHANGE, CUSTOM, ADD TO BLOCK menus and the MACROS can be easily customized to your machine with program MILLCFG which you can run when you exit BOBCAD.

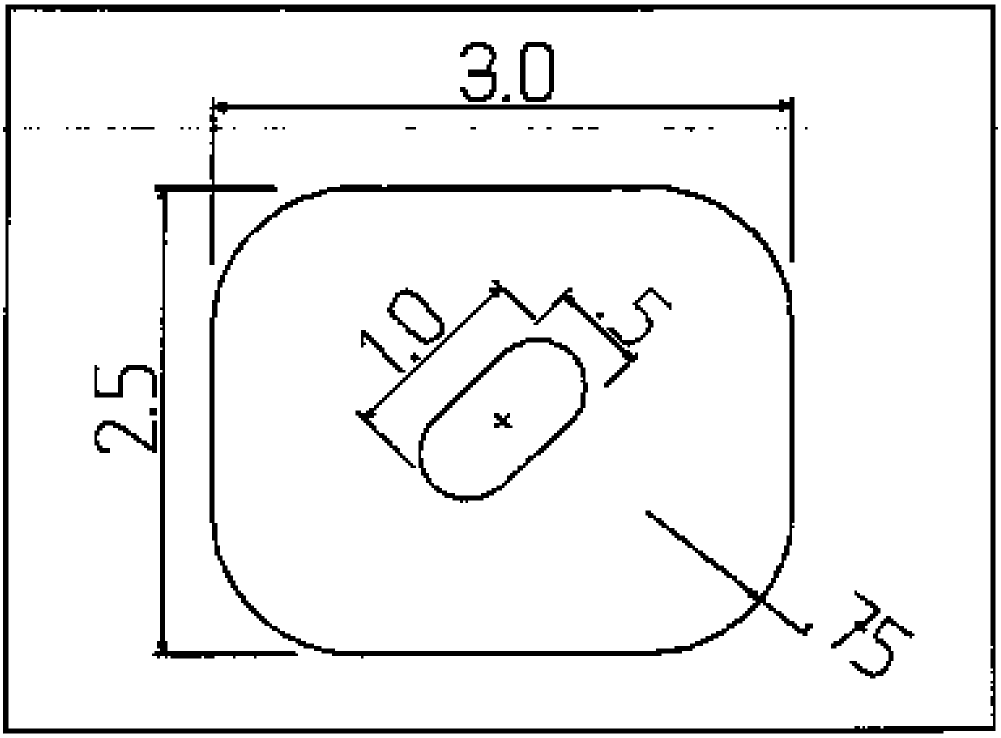
Hit the Q key to Quit BOBCAD. When you quit BOBCAD your NC program will always be saved automatically even if you don't save the drawing!

.

You can run the program **MILLCFG** (just type **MILLCFG** in directory **BOBCAD**) to set BOBCAD to almost any CNC machine. For example **BOBCAD** supports leading zeros, from 2 to 5 digit accuracy behind the period, quadrant circle programming, any tool codes, absolute or incremental programming, Macros, feed rates, block numbering, etc. (see section N.C. CONFIGURATION in this manual).

ROUGHING A PART

Draw this part using the **RECTANGLE** function in the **OTHER CURVES MENU.**



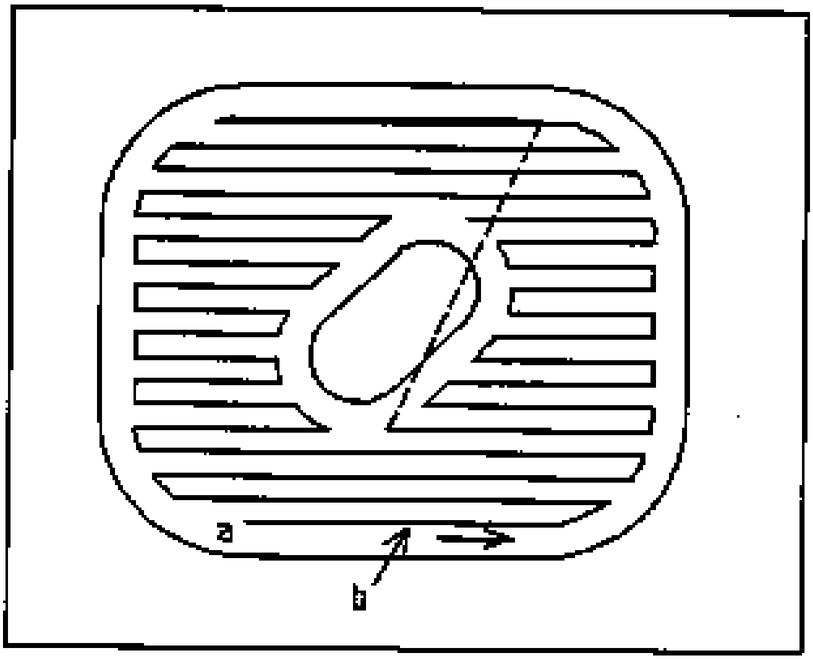
To draw the center position I first drew a box at 0 degrees and then rotated it 45 degrees around the center point using the **ROTATE** function in the **COPY/MOVE MENU.**

To create the roughing path as shown on the next page use function **POCKET** in the **OTHER CURVES MENU.** Select function **AUTO** and pick the outside shape. Answer YES.

> Enter TOOL DIAM = 0.25
> Enter STEP = 0.125
> Enter STOCK = 0.05
>
> (Note: The stock is the amount of material left on the edges which you want to remove in a final pass).

Pick the center shape as the **ISLAND.** Answer **YES.**
Next hit the right button on your mouse for no more islands.

ROUGHING A PART



In the **MILLING MENU** you can now machine the roughing path.

First use function **POINT MOVE** to rapid to location A.

Next go into the TOOL UP/DN MENU and plunge down.

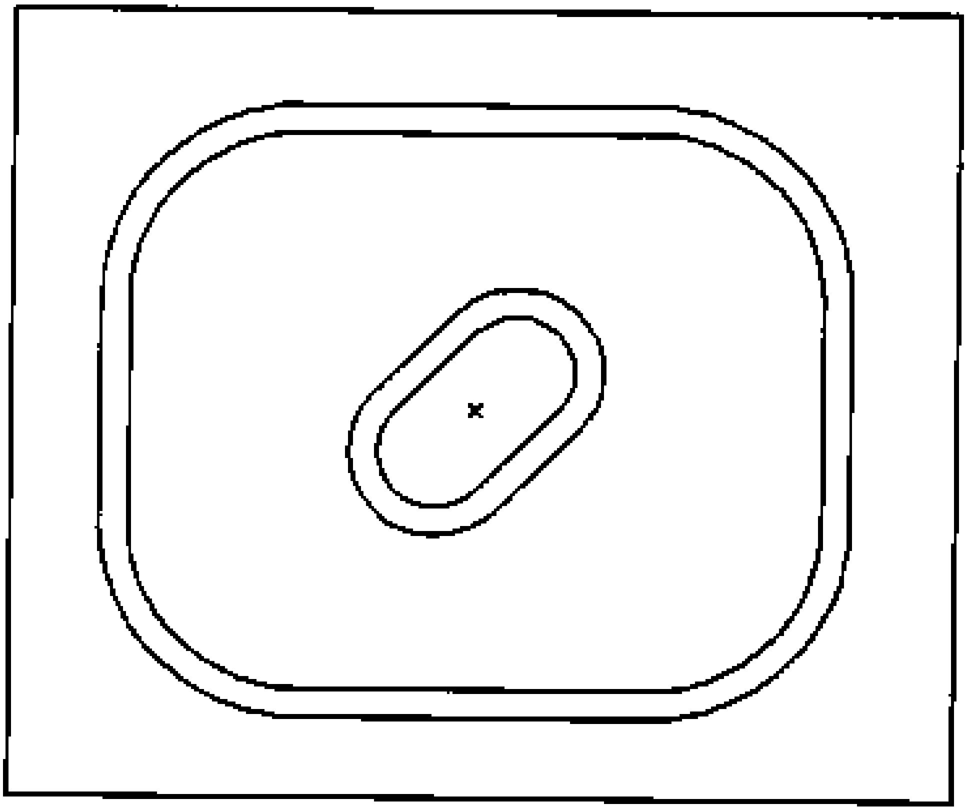
Next use function **AUTO** and pick Line B and direction as shown.

Next hit the] key on your keyboard to machine to the end.

This completes this exercise.

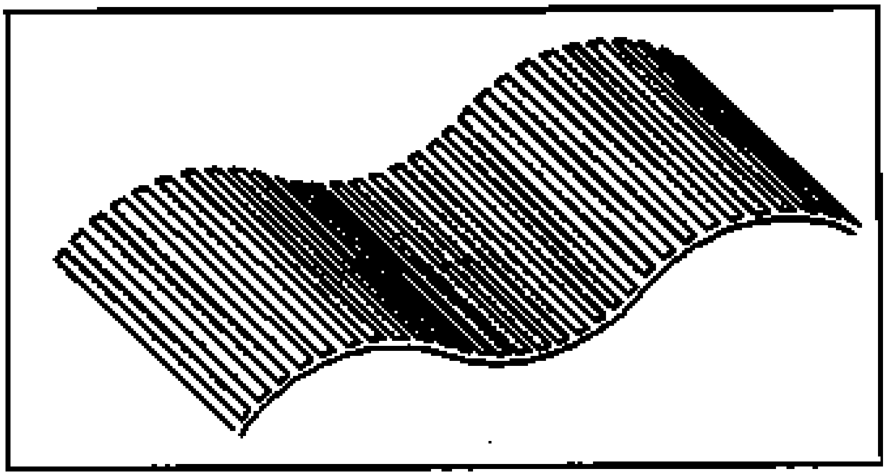
If your N.C. machine has no cutter compensation use the OFFSET function in the OTHER CURVES MENU to create the tool center path as shown below.

In the OFFSET MENU use function CHAIN. Set the offset distance equal to the radius of the tool.



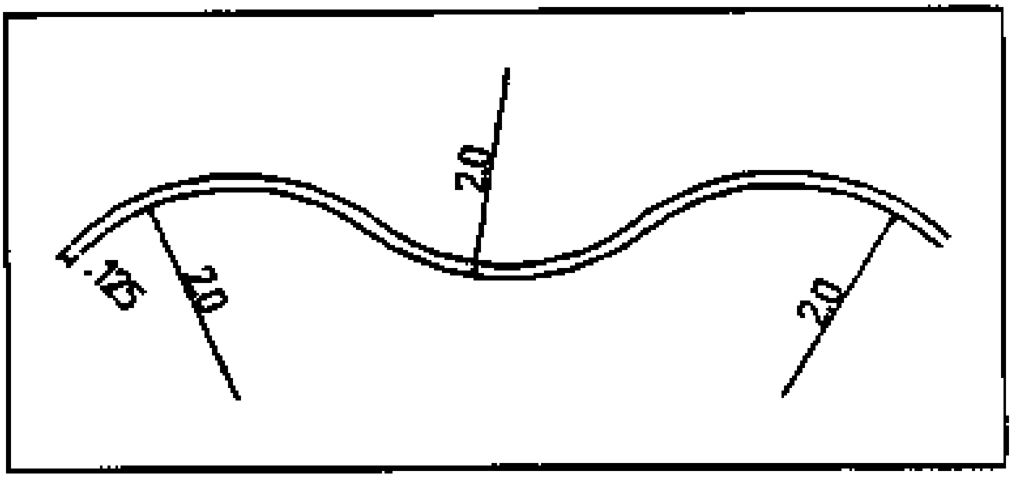
RULED SURFACE
AUTO RULED

This exercise goes through making a ruled surface using the function AUTO RULED in the SURFACE MENU.



START:
Use function ARC: COORDINATE CENTER to draw an Arc at

XC = 0
YC = 0
Radius = 2
Start angle = 0
End angle = 135



Draw a second Arc at :

XC = 5
YC = 0
Radius = 2
Start angle = 45
End angle = 180

Draw a Fillet between the first Arc and the second Arc. Use a radius of 2.0

In order to compensate in Z for a ball-end tool we have to offset this curve as follows:
Select function OFFSET in the OTHER CURVES MENU.

Select CHAIN and enter 0.125 distance for using a 0.25" tool.

Pick the first Arc and indicate direction to the right and the tool above the curve. Next hit the] key to select the whole curve.

Result: an offset curve is drawn which we will use to generate the surface and represent the tool center path. (offsetting is not necessary if your CNC machine can compensate in X Y Z)

Select function THREE-D in the UTILITIES MENU. Next TURN 3-D ON.

Draw a Point at:
X = 0
Y = 0
Z = 0

Select function 7.ROTATE in the COPY/MOVE MENU.

Select option POINT to rotate around.

Select function SINGLE and pick all Arcs.

When all the Arcs are red hit the right button on your mouse. Answer NO to the question "DO YOU WANT TO PICK MORE ENTITIES ?"

Pick the origin Point as the Point to rotate around.

Enter rotation around the AXIS as follows:
X-ANGLE = 90
Y-ANGLE = 0
Z-ANGLE = 0

Result: the curves are rotated from the X-Y plane to the X-Z plane.

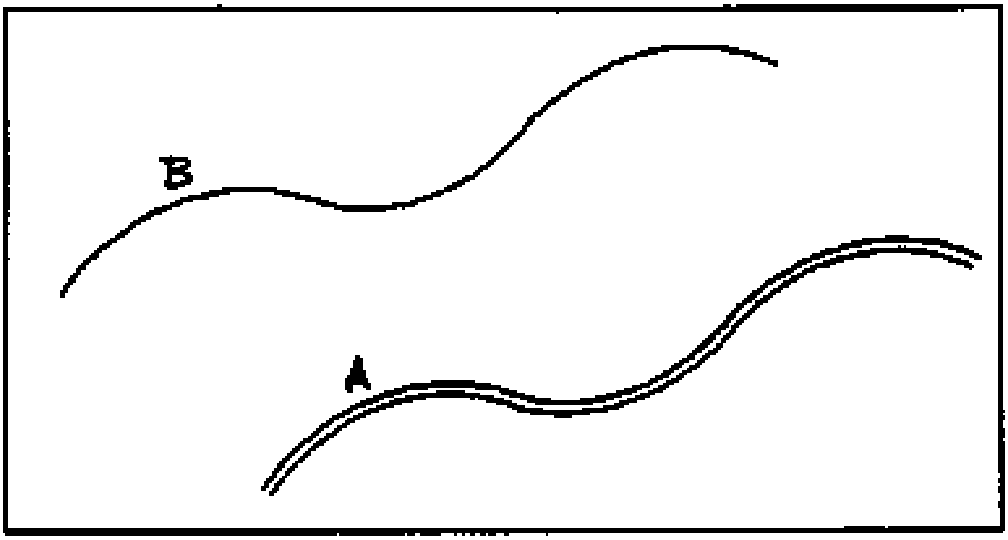
Select function 1.TRANSLATE from the COPY/MOVE MENU.

Select function 4.INC X-Y-Z.

Use function SINGLE to pick the Arcs in the top curve.

Enter: DELTA X = 0
DELTA Y = 5
DELTA Z = 0
COPIES = 1

Result: a second curve is drawn as shown on the next page.



Next: create the surface as follows:
Select function SURFACE in the 3-D MAIN MENU.

Select function AUTO RULED.

Enter number of DIVISIONS = 20.

Pick Arc 'A' as the start of the chain.

Indicate direction to the right and click the left button on the mouse.

Next hit the] key to pick the entire curve.

Pick Arc 'B' as the start of the other chain.

Indicate direction to the right and again pick the entitre curve.

Result: the surface is generated as shown in the beginning of the exercise.

Notes:
The surface is built up of many Lines as if you had drawn them with the LINE MENU functions.
The AUTO RULED function automatically breaks (interpolates) each Arc of each curve into 20 Lines. This is possible because the two curves are identical. In case the two curves are not identical you may not be able to use the AUTO RULED function. The next exercise shows you an example what to do in such a case.

By the way when the Arcs were interpolated into Lines the original Arcs were saved on layer 100 and blanked in case you need them back.

MACHINING THE CARPET

Select function NC.......CAM in the MAIN MENU.

Select function 1.NEW FILE to start a new NC program and enter a file name.

As you can see the tool is up in clearance and function 6 is set to RAPID MODE.

Next select function 3.POINT MOVE and with the cursor pick the start of the carpet in the lower left corner.

Result: A rapid move is generated with the tool still in clearance.

Next select function 7.TOOL UP/DN

Select function Z = FIXED so it changes to Z FROM CURVE.

Select function 4.RAMP and pick the same start point on the carpet.

If FEED RATE is still on RAPID MODE then select FEED MODE now.

Result: the tool moves down. (The part is actually in Z positive)

Next: Select function 5.AUTO and pick the left Line of the carpet.

Indicate direction away from where the tool plunged down and then click the left button on your mouse to indicate that direction.

Next hit the] key on your keyboard (Next to the Enter key)

Result: the tool path is generated.

When you Quit BOBCAD the NC program will be saved on your hard disk under the name that you entered when you went into the NC...CAM menu with the extension .TAP

Save this drawing for the next exercise.

RULED SURFACE

RULED

To start off make sure you have the carpet drawing from the previous exercise still on the screen. If not then draw it again using the previous exercise.

Next: go to the EDIT MENU and select function DELETE.

Select ALL to delete all.

Next select function UNBLANK from the EDIT MENU.

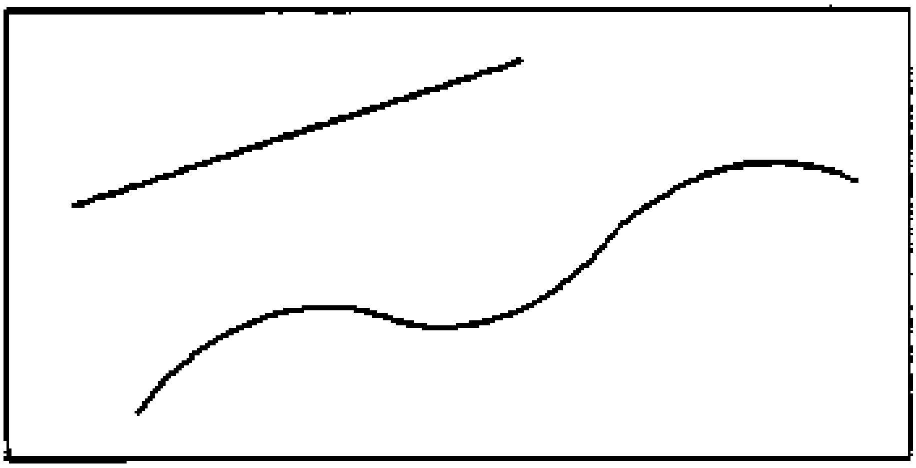
Select ALL to unblank all.

Result: the two wavy curves are back on the screen.

Next delete the second curve so that only the first offset curve remains.

Use function LINE: COORDINATE to draw a Line at:

XS = 0
YS = 5
ZS = 1.25
XE = 5
YE = 5
ZE = 1.25



We will now use the function SURFACE:RULED to draw the ruled surface. For this function the curves have to be interpolated first. Interpolation breaks each Entity into small Entities (small Lines). Do this as follows:

Select function INTERPOLATE from the 3-D MAIN MENU and set number of divisions to 60.

Select SINGLE and pick only the Line as follows:

Pick the Line till it is red.
Next click the right button on your mouse and answer SELECTION OK? YES.

Result: the Line is broken into 60 small Lines. (You can check this out with the VERIFY DATA functions). The original Line is put on layer 100 and blanked.

Next: select function INTERPOLATE from the 3-D MAIN MENU and set number of divisions to 20.

Select SINGLE and pick all the Arcs in the first curve untill they are all red. Next click the right button on your mouse and answer SELECTION OK ? YES.

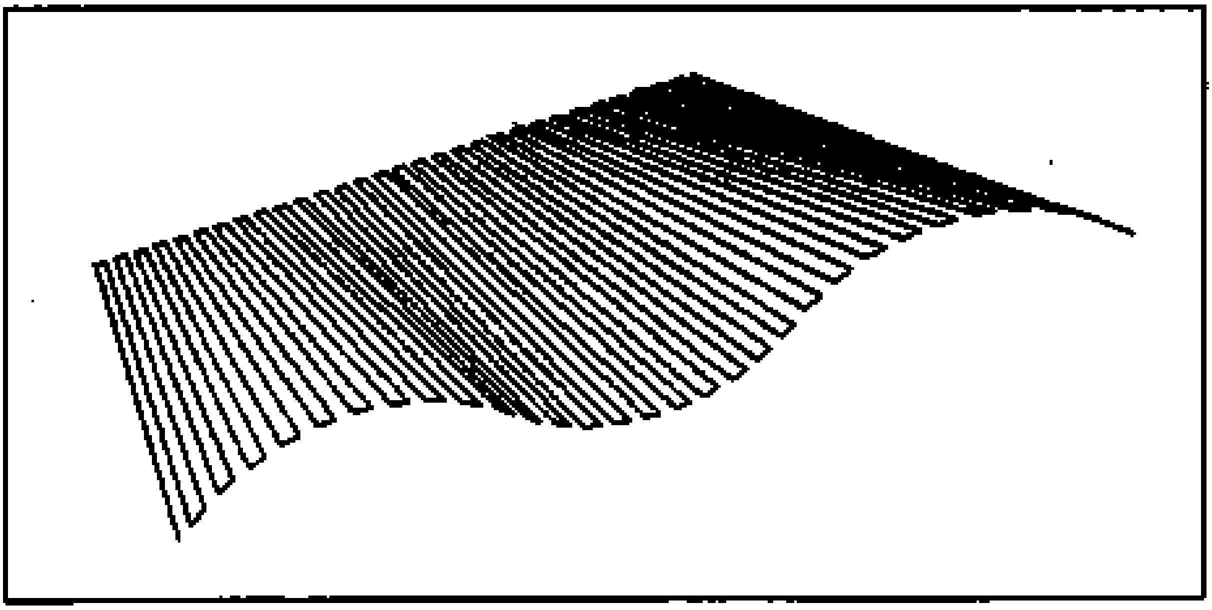
Result: Each Arc is broken into 20 small Lines. (You can again check this out with the VERIFY DATA function). The Arcs are put on layer 100 and blanked for your convenience instead of deleted. This is so that if you made a mistake interpolating or if you need them later you can call them back.

Note that now the first curve has an equal number of Entities to the second curve! This is important: the two curves must have an equal number of Entities.

Next: select function SURFACE from the 3-D MAIN MENU and select function RULED.

Pick the very first Entity on the left end of the wavy curve as the start of the chain (you will see the small Entity turn red). Indicate direction to the right and click the mouse. Next hit the] key on your keyboard to pick the entire curve and answer SELECTION OK ? YES.

Next pick the very first Entity on the left end of the Line so it turns red. Indicate the direction arrow to the right and click the mouse. Next hit the] key on your keyboard to select the entire curve and answer SELECTION OK ? YES. *Result: a ruled surface is generated as shown below.*



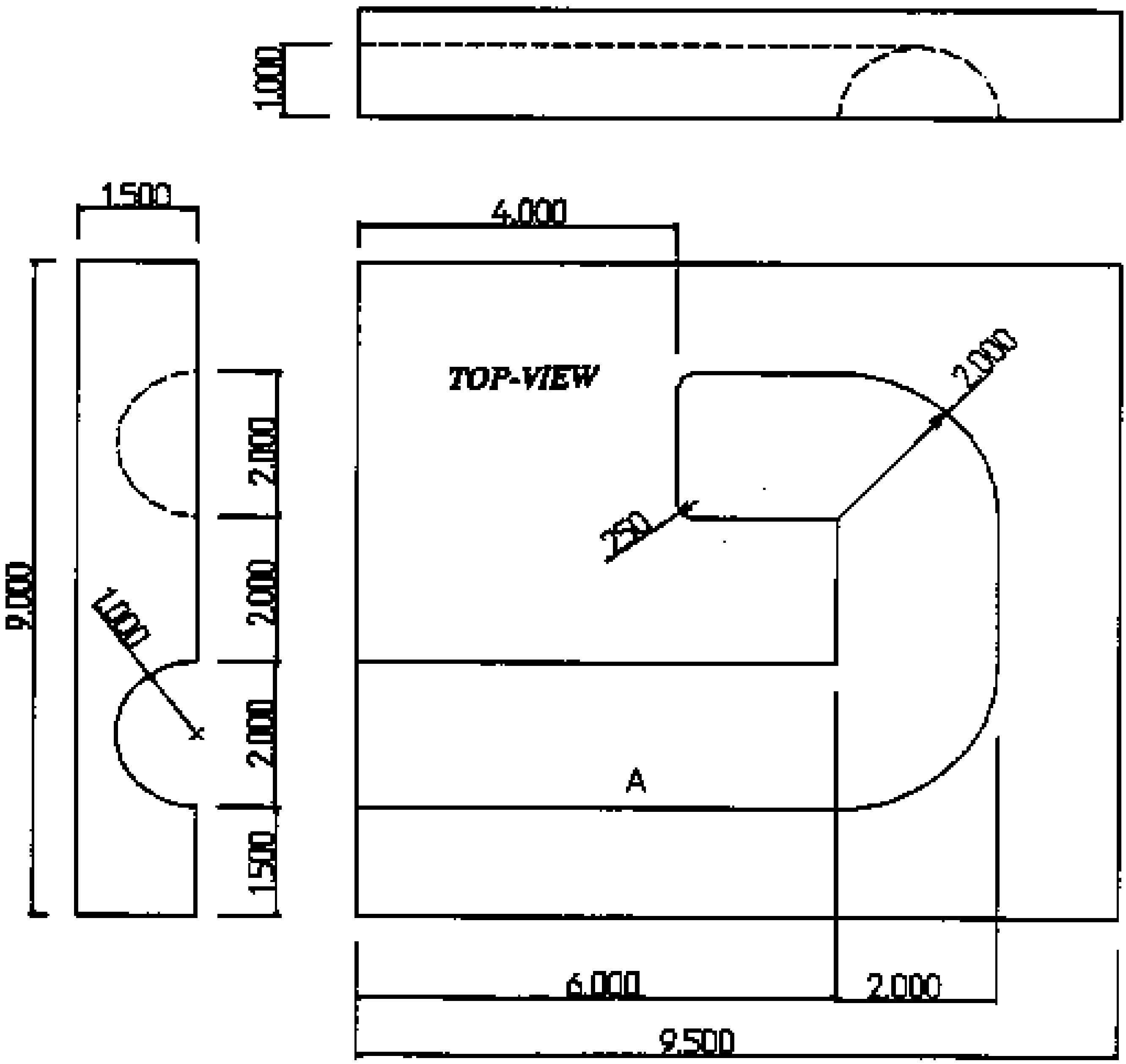
END OF EXERCISE

P-TRAP

This is a 3-dimensional exercise which uses the FOLLOW CURVE function in the SURFACE MENU to generate a surface.

The drawing for the P-trap was submitted to us as shown below. To machine the part you only need to draw the inside curve 'A' in the top-view (see picture below and on next page).

Stay in 2-D mode and only draw outline 'A' in the top-view using your skills from the earlier exercises.



...............P-TRAP CONTINUED

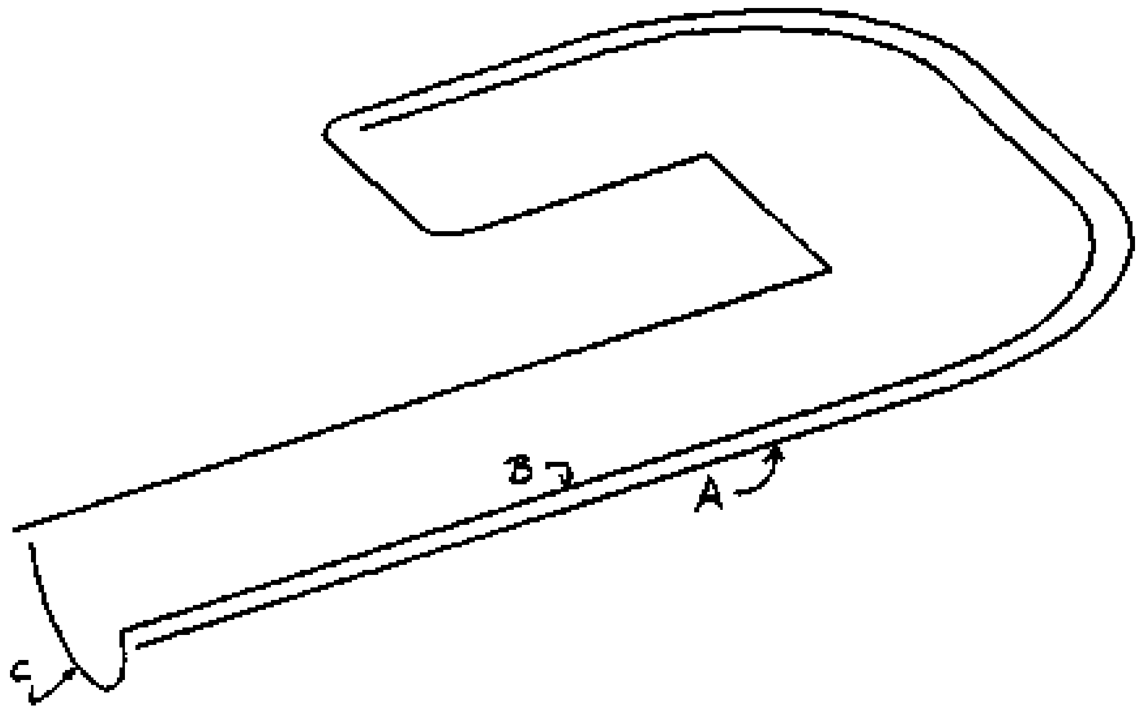
Once you have drawn curve 'A' now use the OFFSET function in the OTHER CURVES MENU to draw curve 'B' using a 1/4" offset. Note that the offset is only along one side of the curve ! (We are using a 1/2" ball-end cutting tool, therefor we have to create a 1/4" offset to compensate for the tool radius).

Next: turn 3-D mode on.

Next: Go to the ZOOM MENU and select function SIDE VIEW in the 3-D views menu.

Next: Draw Arc 'C' with coordinates XC = 0, YC = 1, ZC = 0, RADIUS = 0.75, START ANGLE = 180, END ANGLE = 360. (The P-TRAP radius is 1". Because we are using a 1/2" tool the Arc is drawn with a radius of 3/4" to compensate for the tool radius in Z)

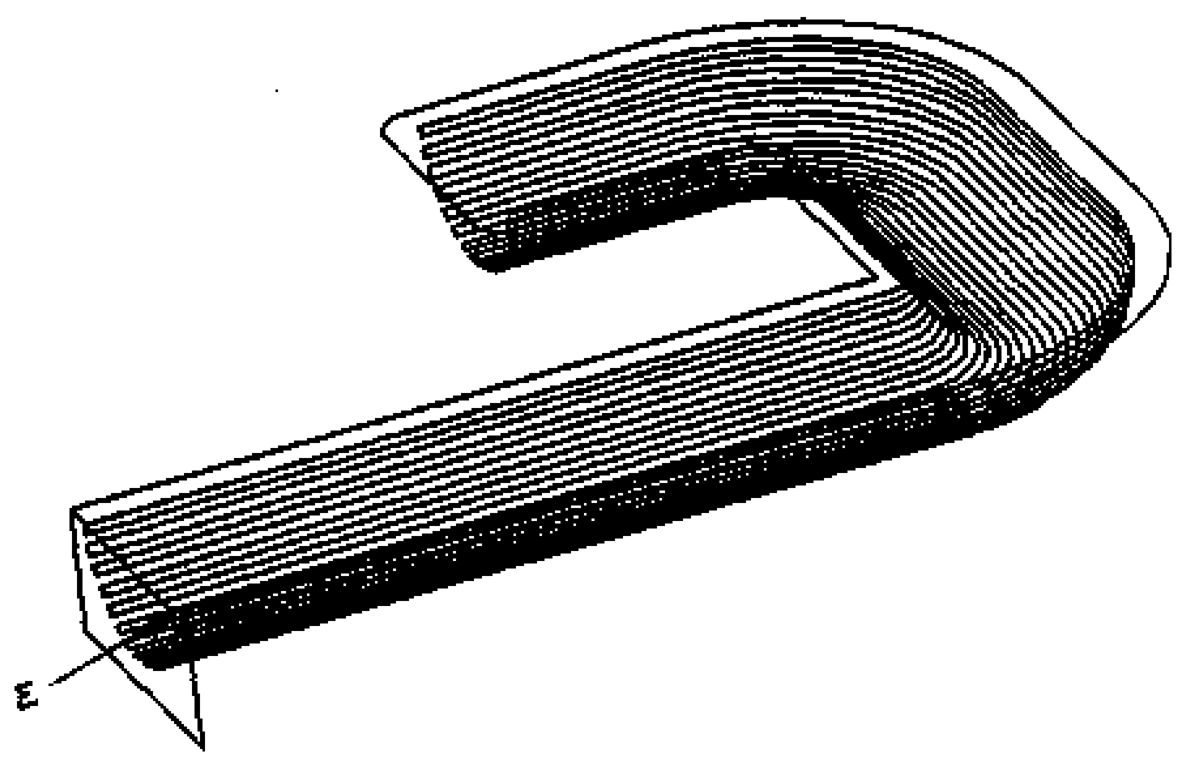
Next: Use the INTERPOLATE function in the 3-D MENU and pick Arc 'C'. Set divisions to 30. (This divides Arc 'C' into 30 Line segments)



............P-TRAP CONTINUED

Use the ZOOM MENU to return to the DEFAULT 3-D view.

Next: Use the LAYER functions to set NEW LAYER to 10.
Next: Go to the SURFACE MENU.
Next: Select function FOLLOW CURVE.
Next: Pick curve 'B' as the curve.
Next: Pick Arc 'C' , (which is now actually 30 Line segments), starting at point 'E' as the path to be followed.
Result: The surface is generated as shown below.



............P-TRAP CONINUED

To machine this shape blank layer 0 first.
Next: Select function NC.....CAM in the MAIN MENU.
Next: Select function 1.NEW FILE to start a new NC program and enter any name (for example TEST) (the name can not have more than 8 characters and no spaces)

As you can see the tool is up in clearance and function 8 is set to RAPID MODE.
Next: Select function 3.POINT MOVE and with the cursor pick the start of the P-TRAP at point 'E' in the lower left corner.

Result: A rapid move is generated with the tool still in clearance.

Next: Select function 7.TOOL UP/DN.
Next: Select function Z=FIXED so it changes to Z FROM CURVE.
Next: Select function 4.RAMP and pick the same start point 'E'.
Note: If FEED RATE was still at RAPID MODE then you will now get a warning. Select FEED MODE.

Result: the tool moves down.

Next: select function 5.AUTO and pick line 'B' of the P-TRAP.
Indicate direction to the right by moving the arrow away from point 'E' untill the arrow points to the right.
Next hit the] key (square bracket) on your keyboard (Next to the Enter key).

Result: the tool path is generated. When you QUIT BOBCAD file TEST is saved automatically on your hard disk under the name TEST.TAP. You can now send it to your machine using GEOTALK or any other communications package. This is described in the manual under the section RS-232 COMMUNICATION.

3-D EXERCISE

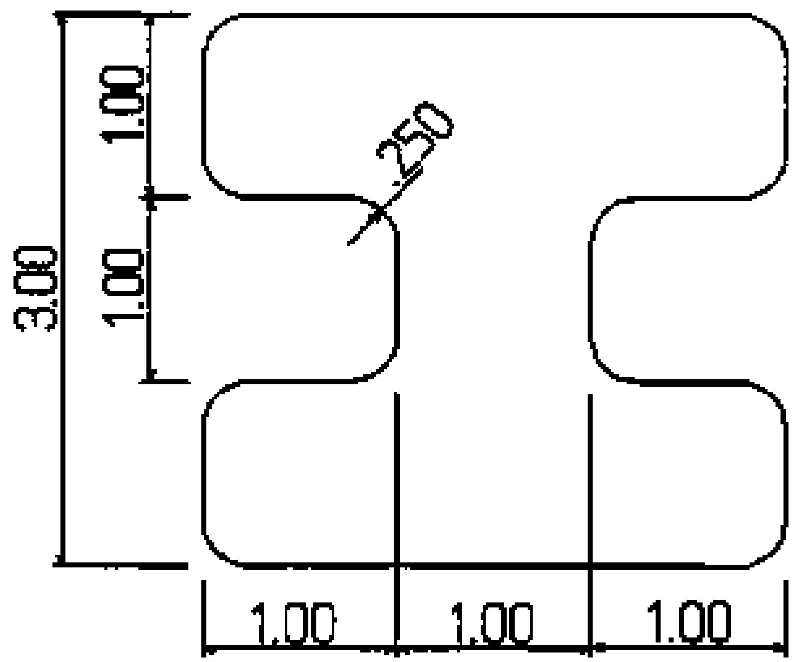
USING FUNCTION: CROSS-SECTION

3-D surfaces can be generated in 3 ways in BOBCAD:

1. Ruled surface between 3-D curves.
2. A curve that follows another curve.
3. Combine a top-view with a cross-sectional view (front view or side view)

This exercise uses the third option.
First we draw the top-view of the part, next we draw the front view (cross-sectional view). Then we create the 3-D surface by adding the two views together.

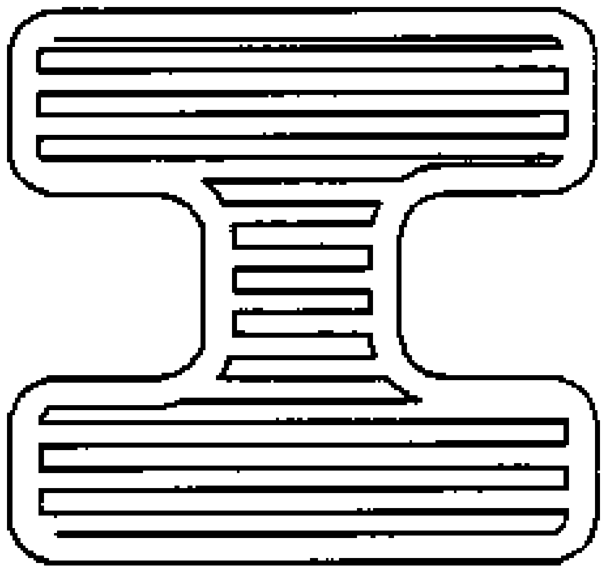
Start: With the skills you learned in earlier exercises draw the top-view of the part as shown below with the lower left corner of the shape at X = 0, Y = 0 (Do not yet switch to 3-D mode untill later). It is not necessary to draw the dimensions.



.........cross-section continued.

Now draw the roughing path as follows:
Select function POCKET in the OTHER CURVE MENU.
Select function AUTO.
Pick the shape which will turn completely red.
Answer selection O.K.: YES (if the whole shape is red).
Next enter TOOL DIA = .25, STEPPING DIST. = 0.12, STOCK DIST. = 0.03 (this will leave 0.03" of stock on the walls and the tool will make passes that cut 0.12" of material at a time)
Result: an offset path is drawn on the inside of the shape.
Next: the prompt on the screen asks for the islands. This shape has no islands so just hit the right button on the mouse.

Result: A roughing shape is drawn as shown below. If this had had a fixed depth you could now machine it using the NC.....CAM functions without going into 3-D.



..........cross-section continued.

The next step is to draw the cross-sectional view.

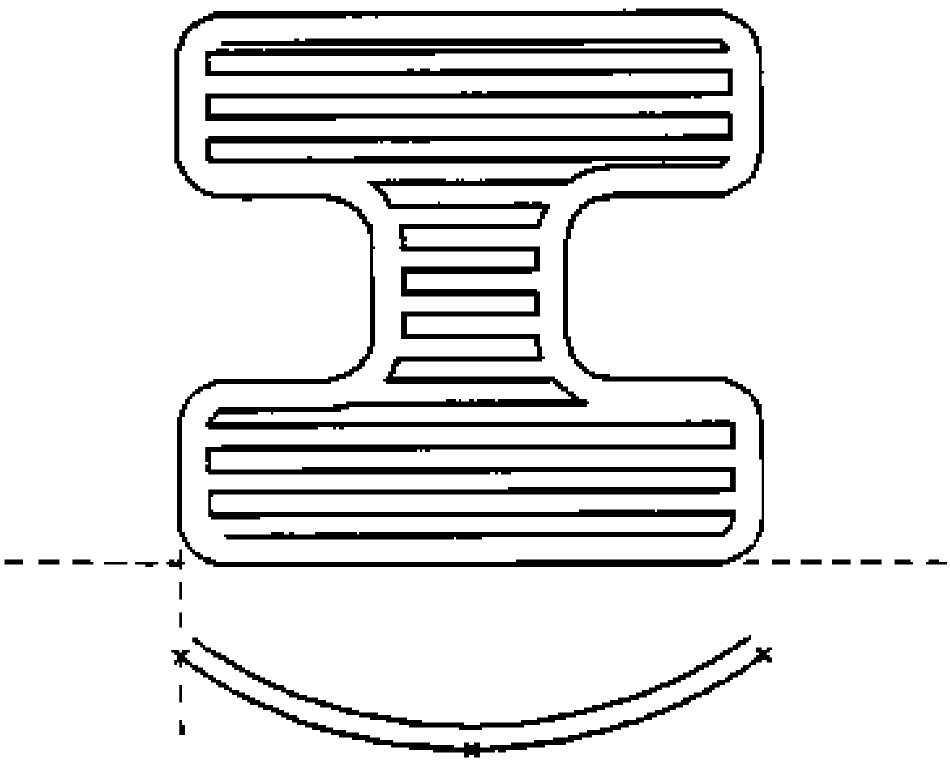
Draw 3 Points below the X-Axis as follows:
Draw a Point with coordinates X = 0, Y = -0.5
Draw a Point at X = 1.5, Y = -1.0
Draw a Point at X = 3, Y = -0.5

Select function 3-ENTITIES in the ARC MENU and pick the first Point, then the second Point and then the third Point.
Result: an Arc is drawn.

We want the bottom of the pocket to have this curve: 1/2" deep at the edges and 1" deep along its center-line.

Next: because we are using a 1/4" ball-end tool and most CNC machines cannot offset in Z, we will now draw a 1/8" (radius of the tool) offset of the Arc as follows:

Select function OFFSET in the OTHER CURVES MENU.
Select function SINGLE.
Enter distance 0.125
Pick the Arc and next indicate the inside.
Result: An offset Arc is drawn which represents the tool center as seen in the front-view.



........cross-section continued

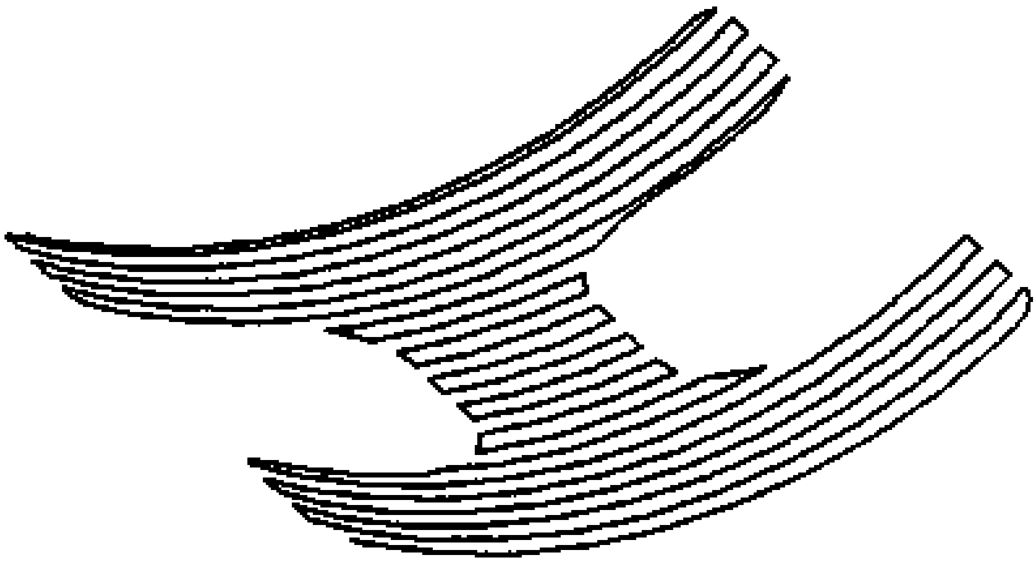
Change the new layer to 10 (in the layer menu) so that the 3-D surface will be drawn on layer 10 which makes it easier later on to separate the parts out.

Select function THREE-D in the UTILITIES MENU.
Next: select function TURN 3-D ON.
Next: Enlarge the view using functions ZOOM: VIEW ALL
Next: select function CROSS-SECTION in the THREE-D MENU.
Next: select function FRONT VIEW.
Next: select function 2.CHAIN.
Pick Line A as the start of the Chain.
Indicate direction to the right.
Next hit the] key (Next to the Enter key) so the roughing path turns red; and answer YES to the question SELECTION O.K ?
Next: pick Arc 'B' in the front-view and indicate direction to the right (doesn't matter in this case as there is only 1 entity in the cross-sectional view.
Next hit the] key and answer YES to SELECTION O.K.?

Result: a 3-D shape is drawn combining the top and front views.

Now blank layer 0 and layer 1000 to layer 1002.
Next hit the R key to Repaint the screen.
The result is shown on the next page.

(Additionally a side view could be added but then modal ABSOLUTE in the 3-D menu needs to be set INCREMENTAL.)

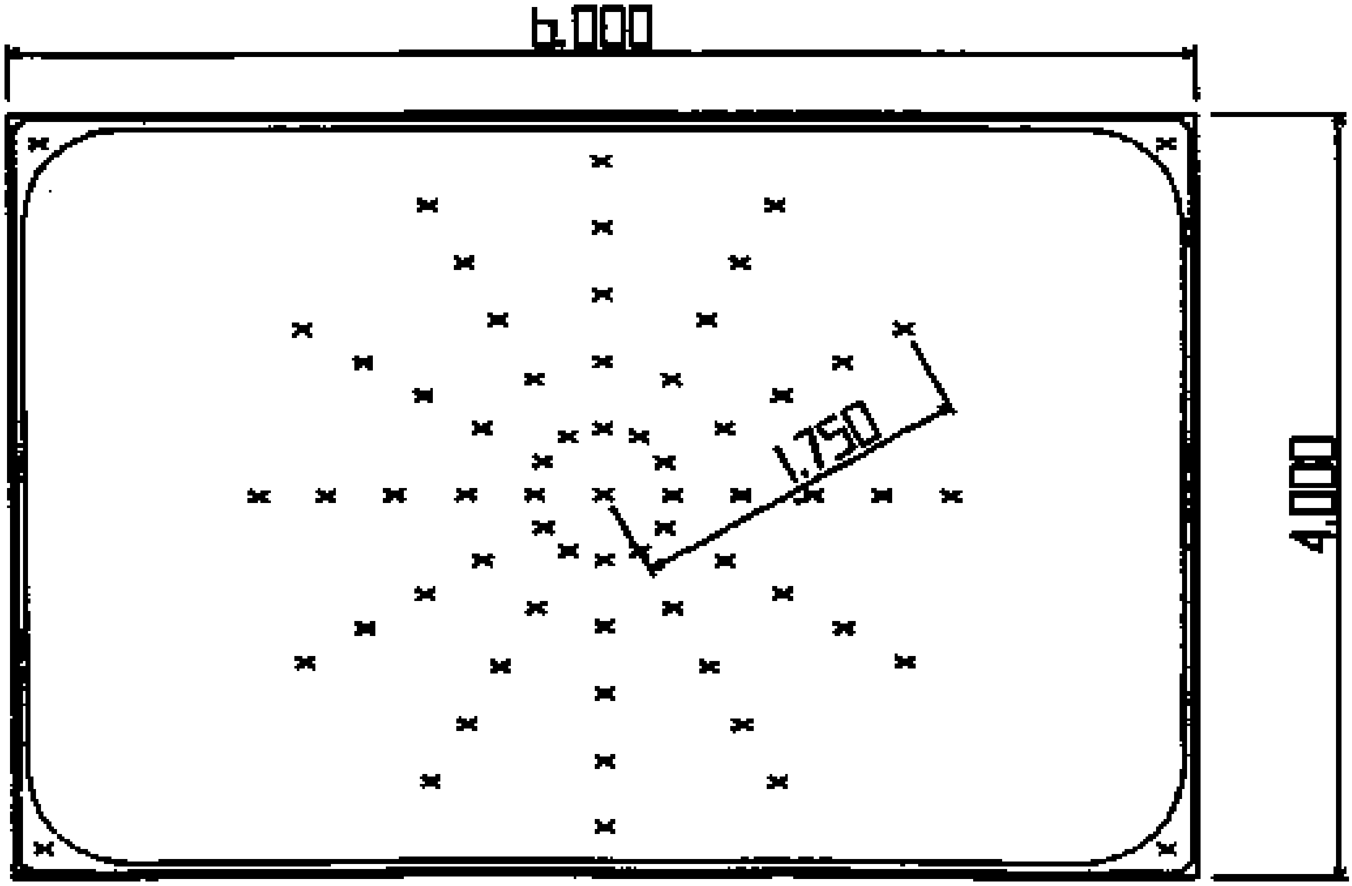


<u>This exercise provided by Gary Svenson of SVENSON MACHINE, Bethel, CT.</u>

The part is shown below and is a 6" x 4" VACUUM BOX, with vent holes and a recess for a 1/16" cover.

THE NC PROGRAM IS SHOWN AFTER THE EXERCISE.

Gary programmed this part for a COMPUMACHINE SERIES 2 CNC MILLING MACHINE using a DYNAPATH DELTA 10 CONTROL. The DYNAPATH has its own canned cycles but for this exercise he used the pocketing features of BOBCAD. Following this exercise is another exercise also written by Gary that uses the canned cycles of the DYNAPATH. He has included a copy of the settings he needs for his DYNAPATH such as the Tool codes, Feed rates etc. These settings are in file MILL25.CFG To set MILL25.CFG to your controller you must run program MILLCFG when you install BOBCAD. This is explained in the manual (see index).



DRAWING PRINTED ON A STANDARD PRINTER

START OF EXERCISE BY GARY SVENSON

-------------------DRAWING THE PART----------------------

1. Function OTHER CURVES
2. Function RECTANGLE
3. Select COORD.
4. Enter X = 0 Y = 0
5. Select Abs. X, Y
6. Enter X = 6 Y = 4
7. Radius = 0

8. Press Z key for ZOOM
9. Function VIEW ALL
10. The rectangle is now enlarged for ease of drawing.

11. Press the O key for OTHER CURVES
12. Function OFFSET
13. Select CHAIN
14. Enter 0.025
15. Follow screen instructions for offsetting chain (keep arrow on the inside of chain) and choose YES for selection OK.
16. Select answer NO for continue offset.

17. Press the A key for ARC MENU
18. Function FILLET
19. Put a .093 Radius in each corner of the inner rectangle.

20. Press the O key for OTHER CURVES
21. Function RECTANGLE
22. Function COORD.
23. Enter X = .075 Y = .075
24. Function ABSOLUTE X,Y
25. Enter X = 5.925 Y = 3.925
26. Radius = .5
    The shape on your screen should now look like the part on the previous page.

27. Select function POINT from MAIN MENU
28. Function COORD.
29. Enter X = .150 Y = .150
30. Press the C key to turn the calculator on.
31. Enter 6 - .15 =

32. Press Enter again to accept the value (X = 5.85)
33. Press Enter to accept default Y = .15

34. Press the M key to return to the MAIN MENU
35. Select function COPY/MOVE
36. Function #1 TRANSLATE
37. Function INC X Y
38. Select SINGLE
39. Pick the two Points at the bottom two corners
40. Press Enter twice.
41. Delta X = 0 Delta Y = 3.7 Copies = 1
42. Now there should be a Point in each of the four corners.

43. Function LAYER from the MAIN MENU
44. Function #1 NEW
45. Enter 1 as new layer number.

----------------DRAWING THE VENT HOLES--------------------

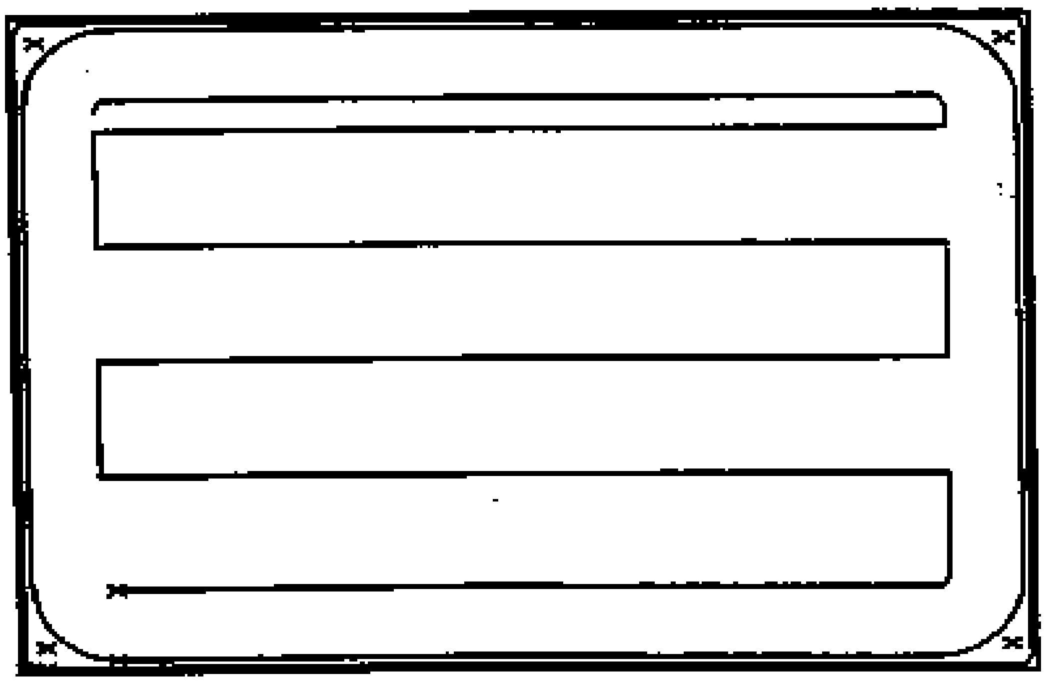
46. Press the P key to go to the POINT MENU.
47. Select COORD.
48. Enter X = 3 Y = 2
49. Press the M key to return to the MAIN MENU
50. Function COPY/MOVE
51. Function #1 TRANSLATE
52. Function #3 INC. X Y
53. Select SINGLE
54. Pick the Point at the center of the pocket.
55. Press Enter twice.
56. Enter Delta X = 0 Delta Y = .35 Copies = 5

57. Function #2 ROTATE
58. Function #2 BY POINTS
59. Select SINGLE
60. Pick the 5 new Points (not the Point at the Pocket center)
61. Press enter twice.
62. Pick the Point at the center of the Pocket as the "FROM" Point.
63. Pick the same Point again as the "TO" Point.
64. Angle = 30 Copies = 11

65. Press the P key for POINT MENU.
66. Select COORD.
67. Enter X = .025 Y = .025

71. Press the E key for the **EDIT MENU**
72. Function **BLANK**
73. Select **LAYER**
74. Enter Layer 1 to Layer 1

75. Press the O key to go to the **OTHER CURVES MENU**
76. Function **POCKET**
77. Function **AUTO**
78. Pick the Inner Pocket as the Outer Shape.
79. Answer **YES**
80. Tool Dia. = .750 Step = .700 Stock dist. = .05
81. Press enter once (for no islands)



82. Press the P key for POINT MENU
83. Select END
84. Pick the bottom left of the roughing tool path.
85. Press Enter
86. Funcion PERPENDICULAR
87. Pick the new Point
88. Pick the Line immediately below the Point.
89. Now with the same function pick the Point at coord X .15 Y .15
90. Pick the Line directly below the Point.

--------------------MACHINING THE PART----------------------

91. Go to the MAIN MENU
92. Function N/C CAM
93. Function MILL 2.5
94. Enter a file name (BOBCAD will add .TAP automatically to the end of the file name)

--------------------CUTTING THE POCKET----------------------

95. Now we will first mill out the main Pocket.
96. Set Z cutting depth to Z = -.375
97. Function point x
98. Rapid to start of pocket roughing path
99. Use the EDIT BLOCK function to read Z0.5 at the end of the line

100. Set RAPID/FEED to FEED
101. Bring tool down
102. Function ADD TO BLOCK and select your feedrate.
103. Function MAIN MENU and then function auto, and pick the first line of the roughing path and direction to the right.
104. Pick the last Entity (which is the last Line of the pocket path)
105. Set RAPID/FEED to RAPID
106. Set clearance to Z = 0.05
107. Bring Z axis back up to clearance
108. Function 'point x' to rapid back to lower left of pocket path
109. Set to FEED and bring cutter down.

110. Pick G41 to turn cutter comp. to the left of the path
111. Select function 'point x'
112. Move to the Point directly below where you are now
113. Function 'auto'
114. Indicate the bottom of the pocket for a clean up cut

115. Direction should be to the right
116. Last Entity on the left of the start point
117. Feed cutter back up
118. Function G40 to turn cutter comp. off

--------------------CUTTING THE RECESS------------------------
119. Function **MAIN MENU**
120. Function **TOOL CHANGE**
121. Select T2 (3/16") End Mill
122. Select M3 for spindle on
123. Set to **RAPID**
124. Set depth to Z = -.062
125. Function 'point x'
126. Pick the Point at coord. X .15 Y .15
127. Edit in at the end of the block Z.05
128. Set to **FEED**
129. Bring tool down
130. Set G41
131. Function point x
132. Pick the Point directly below your present position (Press the P key to make sure you select the Point')

133. Function **AUTO**
134. Pick the Line that you are on.
135. Direction to the right
136. Last Entity is the Fillet next to the start Point
137. Feed tool up

138. Set to **RAPID**
139. Select G40 to turn cutter comp. off

----------TOOL CHANGE FOR DRILLING THE HOLES----------
140. Function **MAIN MENU**
141. Function **TOOL CHANGE**
142. Select T3 (1/8" Drill)
143. Select M3 to turn spindle on

-------RETURN TO CAD TO UNBLANK THE VENT HOLES----------
144. Press the Q key to return to CAD
145. Press the E key for EDIT MENU
146. Function **UNBLANK**
147. Function **ALL** (Hole pattern should reappear)

------------------DRILLING THE HOLES------------------------

148. Press The N key to return to NC...CAM and resume with LAST FILE
149. Set cutting depth to Z = -.4
150. Function DRILL
151. Function G81
152. Pick the 4 corner holes
153. Press Enter
154. Set cutting depth to Z = -.6
155. Function G81
156. Pick any hole in vent hole pattern
157. Press Enter, then EDIT your Z ref. to -.325
158. Function DRILL
159. Function G81
160. Continue drilling the rest of the vent holes
161. Press Enter
162. Press the Q key to end and save the NC program

163. The NC program is now saved on your disk. The name of the file is the name you entered at the beginning. BOBCAD automatically adds .TAP to the end of the name.
164. Now exit BOBCAD and send it to your N.C. machine.

NAME: A.TAP 16:52:30 05-26-1990

```
(A)
N1M3T1
N2M3E0
N3G70
N4G90
N5G80
N6G00X.5750Y.5000Z.05
N7G01Z-.3750F10
N8X5.4250
N9G03X5.5000Y.5750I5.4250J.5750
N10G01Y1.2000
N11X.5000
N12Y1.9000
N13X5.5000
N14Y2.6000
N15X.5000
N16Y3.3000
N17X5.5000
N18Y3.4250
N19G03X5.4250Y3.5000I5.4250J3.4250
N20G01X.5750
N21G03X.5000Y3.4250I.5750J3.4250
N22G00Z.0500
N23G00X.5750Y.5000
N24G01X.5750Y.5000Z-.3750
N25G41
N26G01Y.0750
N27X5.4250
N28G03X5.9250Y.5750I5.4250J.5750
N29G01Y3.4250
N30G03X5.4250Y3.9250I5.4250J3.4250
N31G01X.5750
N32G03X.0750Y3.4250I.5750J3.4250
N33G01Y.5750
N34G03X.5750Y.0750I.5750J.5750
N35G01Z.0500
N36G40
N37M6T2
N38M3E0
N39G00X.1500Y.1500Z.05
N40G01Z-.0620F10
N41G41
N42G01Y.0250
N43X5.8820
N44G03X5.9750Y.1180I5.8820J.1180
N45G01Y3.8820
N46G03X5.8820Y3.9750I5.8820J3.8820
N47G01X.1180
N48G03X.0250Y3.8820I.1180J3.8820
N49G01Y.1180
N50G03X.1180Y.0250I.1180J.1180
N51G01Z.0500
N52G40
N53M6T3
N54M3E0
N55G01X.1500Y.1500Z-.4000
```

```
N56X6.8500
N57G81X5.8500Y3.8500
N58X.1500
N59X3.0000Y3.7500Z-.6000R-.325F10
N60Y3.4000


NAME: A.TAP                                          16:52:33   05-26-1990

N61Y3.0500
N62Y2.7000
N63Y2.3500
N64Y2.0000
N65Y1.6500
N66Y1.3000
N67Y.9500
N68Y.6000
N69Y.2500
N70X2.1250Y.4845
N71X2.3000Y.7876
N72X2.4750Y1.0907
N73X2.6500Y1.3938
N74X2.8250Y1.6969
N75X3.3031Y2.1750
N76X3.6062Y2.3500
N77X3.9093Y2.5250
N78X4.2124Y2.7000
N79X4.5155Y2.8750
N80X3.8750Y3.5155
N81X3.7000Y3.2124
N82X3.5250Y2.9093
N83X3.3500Y2.6062
N84X3.1750Y2.3031
N85X2.6969Y1.8250
N86X2.3938Y1.6500
N87X2.0907Y1.4750
N88X1.7876Y1.3000
N89X1.4845Y1.1250
N90X1.2500Y2.0000
N91X1.6000
N92X1.9500
N93X2.3000
N94X2.6969Y1.8250
N95X2.6500Y2.0000
N96X3.3500
N97X3.7000
N98X4.0500
N99X4.4000
N100X4.7500
N101X4.5155Y1.1250
N102X4.2124Y1.3000
N103X3.9093Y1.4750
N104X3.6062Y1.6500
N105X3.3031Y1.8250
N106X2.6969Y2.1750
N107X2.3938Y2.3500
N108X2.0907Y2.5250
N109X1.7876Y2.7000
N110X1.4845Y2.8750
N111X2.1250Y3.5155
N112X2.3000Y3.2124
N113X2.4750Y2.9093
N114X2.6500Y2.6062
N115X2.8250Y2.3031
```

```
N117X3.3500Y1.3938
N118X3.5250Y1.0907
N119X3.7000Y.7876
N120X3.8750Y.4845
N121M30
```

NAME: A.TAP 16:52:37 05-26-1990

E

Dear Bob,

This is an exercise for BOBCAD using codes for the Dynapath Delta 10 control, as you requested.

The part is a mounting block for transistors. Some of the actual holes in the original blueprint were omitted because I feel that it would be too repetitious.

I tried to include as many canned cycle codes as possible such as:

Event type (3) is a frame mill where you can mill an inside contour or an outside contour or a pocket.

X = pocket width Y = pocket height Z = Z depth

W = reference distance above part. R = corner radius

D = direction of cut 1 = rough counter clockwise and finish counter clockwise.

C1, is the same as G41

G7 is the code for pocket mill, G0 would be a frame cycle

K = incremental peck distance

P = plunge feedrate F = feedrate L/ = incremental finish pass

V = finish feedrate and $ = conversational event

Event type (4) is a Circle pocket or frame

Event type (5) is a Bolt circle

Event type (6) is a Repeat cycle

For the Repeat Event X/ = incremental X move

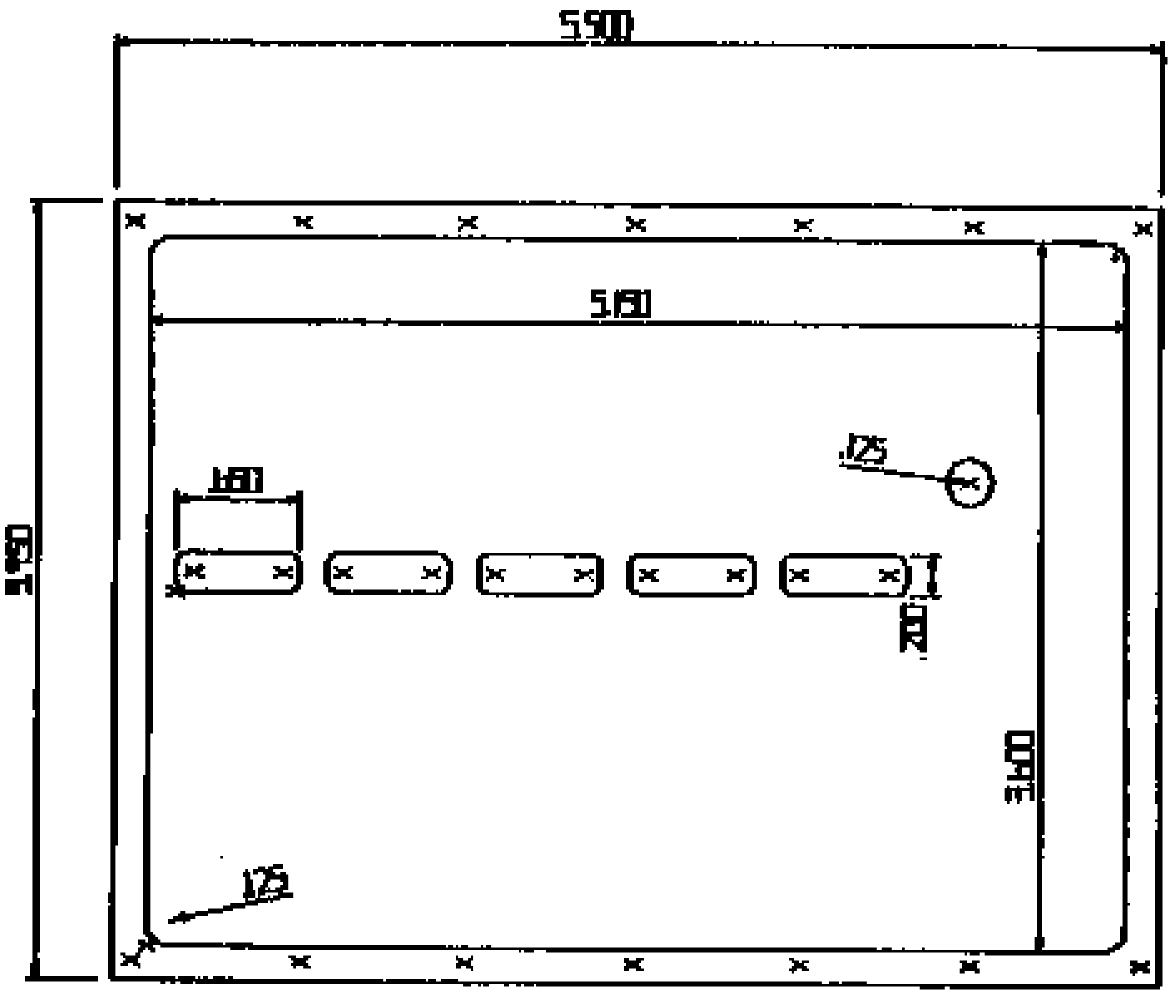
Y/ = incremental Y move (if no / was programmed it would be an absolute move. F = first block to be repeated. T = how many times to be repeated.

Event type (7) FL$ is a subroutine call F= first L = last

Bob I realise this is a basic program but it should get the idea across on how the BOBCAD and Dynapath can work together with their own Custom codes.

Sincerely

Gary Svenson



TRANSISTOR MOUNT

1. Pick OTHER-CURVES by pressing the"O" key
2. Pick RECTANGLE
3. Pick COORD
4. Enter X=0 Y=0
5. Pick Abs. X,Y
6. Enter X= 5.5 Y=-3.95
7. Radius = 0
8. Press the "Z" key for Zoom
9. Pick Window
10. Indicate the bottom & upper right corners
11. Press the "O" key for Other Curves
12. Pick Offset
13. Pick Chain
14. Enter .175
15. Pick any line
16. Stay on the inside of the chain
17. Direction clockwise
18. Hit the] key on your keyboard.
19. Selection is o.k. pick Yes
20. Pick No for Continue Offset
21. Press the "A" key for ARC MENU.
22. Select Fillet
23. Pick any inside corner

24. Enter Radius of .38

25. Repeat procedure for the remaining 3 corners

26. Return to the Main Menu

27. Press the "P" key for Point

28. Pick Coord.

29. Enter X =.100 Y +-.100

30. Return to the Main Menu

31. Pick Copy/Move

32. Pick #1 Translate

33. Pick INC. X Y

34. Pick Single

35. Select Point at the upper left corner

36. Press Enter once and answer NO.

37. Delta X= .883 Delta Y= 0 # Copies = 6

38. From Copy/Move menu choose #1 again

39. Choose #3 INC. X Y

40. Pick # 1 Single

41. Select top 7 points

42. Press Enter once and answer NO.

43. Enter Delta X=0 Delta Y= 3.750 Copies = 1

44. Press the "O" key

45. Pick Rectangle

46. Pick Coord.

47. Enter X= .325 Y= -1.975

48. Pick #2 Abs. X Y

49. Enter X=.975 Y= -1.775

50. Radius .063

51. Press the "P" key

52. Pick Coord.

53. Enter X = .414 Y =-1.875

54. Enter X = .886 Y= -1.875

55. Enter X .325 Y -1.975

56. Enter X .175 Y -3.375

57. Return to Main Menu

58. Pick Copy/Move

59. Pick #1 Translate

60. Pick #3 Region

61. Indicate the top & Bottom corners of the latest pocket

62. Press Enter

63. Delta X = .8 Delta Y = 0 # of copies = 4

64. Press the "P" key

65. Pick Coord

66. Enter X =4.5 Y =-1.4

67. Press the "A" key for Arc

68. Pick Point Center

69. Pick last point

70. Enter Radius = .125

71. Press Enter key twice to accept 0 start angle, 360 end angle.

72. Press the "N" key

73. Pick #1 Milling

74. Select Function POINT.

75. Choose the point at the bottom left corner of the large pocket

76. Pick Function CUSTOM

77. Choose Frame Mill (3)

78. Go to Edit Block from the Main Menu

79. Edit Block as follows

    X5.15Y3.55Z-.37W.05R.38D1C1G7K.3P4F10L/.002V20$

80. Pick Function TOOL CHANGE

81. Pick M6T2

82. Pick MM3E0 from the TOOL CHANGE MENU (e0 is a fixture offset)

83. From MAIN MENU pick POINT X.

84. Pick the Point at the bottom left corner of the small pocket

85. Pick Function CUSTOM

86. Pick (3) frame mill

87. From the main menu pick Edit block

88. Edit block as follows

X.65Y.2Z-.4W-.3R.063D1C1G7P5F10L/.2225V20$

89. Pick Function CUSTOM.

90. Pick (6) Repeat Event

91. Edit to read X.8Y/0Z/0F12T4$

92. Pick Point

93. Pick the point at the center of the 1/4" circle

94. Pick Function CUSTOM.

95. Pick (4) Circle Millt

96. Edit to read

X/0Y/0Z-.67W-.3R.125D1C1G7K.12P4F7L/.001V10$

97. Pick Function TOOL CHANGE.

98. Pick M6T3

99. Pick M3E0

100. Z depth should be set to -.250

101. Pick Function Point X

102. Move to point at the top left corner of the part

103. Pick Drill

104. Pick G81

105. Press Enter

106. From Main Menu pick Edit Block

107. Edit to read G81Z-.250R.05F10

108. Pick Custom

109. Pick (6) Repeat Event

110. Edit to read X/.883Y/0Z/0F19T654$

111. Pick Custom

112. Pick (6) Repeat Event

113. Edit to read X.1Y/-3.75Z/0F20T1$

114. Pick Drill

115. Pick G80

116. Pick G83, (Z should be set to -.6)

117. Pick the point in the small pocket

118. Press Enter

119. Pick Add to block

120. Pick K.1Q.005 then pick F10

121. Go to Edit Block and add R-.3 to the end of the block

122. Pick Drill, G83 then continue drilling the rest of the holes in the small pockets

```
(D^^NABOB)
     T1
  .M6E0
N3G70
N4G90
N5G80
N7G00X.1750Y-3.7750
N8(3)X5.15Y3.55Z-.37W.05R.36D1C1G7K.3P4F10L/.002V20$
N9M6T2
N10M3E0
N11G00X.3250Y-1.9750
N12(3)X.65Y.2Z-.4W-.3R.063D1C1G7P5F10L/.0005V20$
N13(6)X/.8Y/0Z/0F12T4$
N14G00X4.5000Y-1.4000
N15(4)X/0Y/0Z-.67W-.3R.125D1C1G7K.12P4F7L/.001V10$
N16M6T3
N17M3E0
N18G00X.1000Y-.1000
N19G81Z-.2500R.05F10
N20(6)X/.883Y/0Z/0F19T6$
N21(6)X.1Y/-3.75Z/0F20T1$
N22G83X.4140Y-1.8750Z-.6000K.1Q.005F10R-.3
N23X.8860
N24X1.2140
N25X1.6860
N26X2.0140
Y   X2.4860
    X2.8140
N29X3.2860
N30X3.6140
N31X4.0860
N32M30
E
```

Dynapath Setup

CNCCADM : FILE MILL25.CFG

***** FORMAT *****

ABSOLUTE COORDINATES
PERIOD WITH 4 DECIMALS : 12.34567 = 12.3457
NO LEADING 0'S : X123
NO '*' AT END OF LINE
AUTOMATIC LINE NUMBERS: N100

CIRCULAR: X Y I J
(G90 ONLY) CIRCLE: ABS X Y , ABS I J
(G02 - G03) : FULL CIRCLE POSSIBLE

NO SPACING : G01X123Y123
OUTPUT CHANGED COORDINATE ONLY (INCREMENTAL) Y12.345
2. G CODES: G01, G02, G03 (MODAL)
NO SLASH FOR RAPID MOVE

USE ARROW KEYS ! NEXT PAGE = N SAVE = S QUIT = Q

START BLOCK NUMBERING WITH # : 1
INCREMENT BLOCK NUMBERING WITH : 1

***** PARAMETERS *****

YOU MUST USE A PERIOD (.) IN YOUR VALUE !

DEFAULT CLEARANCE HEIGHT: Z = .05000
DEFAULT CUTTING DEPTH Z = .00000

USE ARROW KEYS ! NEXT PAGE = N SAVE = S QUIT = Q

**** FIRST FEW LINES ****
1. (
2. M3T1
3. M8E0
4. G70
5. G90
6. G80

START AUTOMATIC LINE NUMBERING AT LINE: 2

```
**** LAST FEW LINES        ****
1. M30
2. E
3.
4.
5.
6.

NO AUTOMATIC LINE NUMBERING FROM LINE : 2


USE ARROW KEYS        NEXT PAGE = N        SAVE = S        QUIT = Q

**** ENTER CUSTOM BLOCS        ****
1. (3)XYZWRDCGKPFLV$
2. (4)XYZWRDCGKPFLV$
3. (5)X/OY/OZARFGWLH$
4. (6)XYZFT$
5. (7)FL$
6. G90
7. G91
8. 40 CHARACTERS MAXIMUM PER BLOCK
9. M8
10.M9

**** ENTER TOOL CHANGE CODE ****
1. G80
2. M6T2
3. M6T3
4. M6T4
5. M6T5
6. M6T6
7. M6T7
8. M6T8
9. M6T9
10.M30
USE ARROW KEYS        N = NEXT PAGE        SAVE = S        QUIT = Q

** ENTER CODE TO BE ADDED TO END OF BLOCK (for example FEED RATES) **
1. F1
2. F3
3. F5
4. F10
5. F20
6. F50
7. F100
8. K.05Q.005
9. K.1Q.005
10.K.2Q.005


USE ARROW KEYS                         SAVE = S        QUIT = Q
```

N.C. CONFIGURATION

Section 4

Configure BOBCAD to your CNC machine

RS-232 COMMUNICATION

N.C. CONFIGURATION

In the CAM section of BOBCAD, one of the first things the user is prompted for is which configuration file to use. BOBCAD provides up to eight of these files, allowing the user to customize the program output for up to eight different machines.

For example, a shop may have one CNC lathe and one milling machine. The lathe has an older Yasnac control that does not use a decimal point and cannot use an "R" for radius designation. The mill, on the other hand, has a more modern controller that does use a decimal and also can be programmed with an "R" for G02 and G03 moves.

BOBCAD can easily handle these two machines by having two different configurations. The number one file can be set up for the lathe and the number two file can be set for the mill.

Go through the following steps in **MILLCFG** to set BOBCAD to your controller.

<u>RUNNING MILLCFG</u>
Return to **DOS** (Disk Operating System) and go to the **BOBCAD** directory. Next type **MILLCFG** to start the configuration program. Select an NC Configuration File such as CAM-1.CFG

The following page appears on your screen:

FILE CAM-1.CFG

******* FORMAT *******

(1) ABSOLUTE COORDINATES
(2) PERIOD WITH 4 DECIMALS : 12.3457
(3) NO LEADING 0'S : X123
(4) NO LINE NUMBERS

(5) CIRCULAR: X Y I J
(6) (G90) ONLY! CIRCLE: ABS X Y, INC I J
(7) (G02-G03) : FULL CIRCLE POSSIBLE

(8) LEAVE SPACING : G01 X123 Y123
(9) OUTPUT ALL COORDINATES (INCREMENTAL) X0.000 Y12.345
(10) 1. G CODES: G01, G02, G03 (SHOW ALL)
(11) NO SLASH FOR RAPID MOVE

Use the Up - Down Arrow keys on your keyboard to move the Modal that you want to change.
Use the Left - Right Arrow keys on your keyboard to change that Modal.

__Explanation of MODAL OPTIONS__

(1) ABSOLUTE COORDINATES
INCREMENTAL COORDINATES

(2) OUTPUT WITH PERIOD, 2 DIGIT ACCURACY (123.12)
OUTPUT WITH PERIOD, 3 DIGIT ACCURACY (123.123)
OUTPUT WITH PERIOD, 4 DIGIT ACCURACY (123.1234)
OUTPUT WITH PERIOD, 5 DIGIT ACCURACY (123.12345)

OUTPUT WITHOUT PERIOD, 2 DIGIT ACCURACY (12312)
OUTPUT WITHOUT PERIOD, 3 DIGIT ACCURACY (123123)
OUTPUT WITHOUT PERIOD, 4 DIGIT ACCURACY (1231234)
OUTPUT WITHOUT PERIOD, 5 DIGIT ACCURACY (12312345)

(3) NO LEADING ZEROS :X123
LEADING ZEROS : X+000123

Note: When you use Leading zeros do not set modal (2) to output with period.

(4) NO !* AT END (of each block)
!* AT END (of each block)

(5) NO LINE NUMBERS (at front of each block)
AUTOMATIC LINE NUMBERS: N100 (at front of each block)

(6) CIRCULAR: X Y I J (Most controllers use this option)
CIRCULAR: I J R C

(7) (G90 ONLY) CIRCLE: ABSOLUTE X Y , INCREMENTAL I J
(G90 ONLY) CIRCLE: ABSOLUTE X Y , ABSOLUTE I J

Note: This Modal applies only when you program in absolute coordinates. (Most controllers use the first option)

(8) (G02 - G03) : FULL CIRCLE POSSIBLE
(G02 - G03) : MAXIMUM 1/4 CIRCLE

Note: If your controller does not program more than a 90 degree Arc just must set this modal to MAXIMUM 1/4 CIRCLE

(9) LEAVE SPACING : G01 X123 Y123
NO SPACING : G01X123Y123

Note: Set this modal to NO SPACING to remove the blanks If you want a smaller program.

(10) OUTPUT ALL COORDINATES
OUTPUT CHANGED COORDINATES ONLY

Important: TAB SEQUENTIAL PROGRAMMERS: This modal must be set to OUTPUT ALL COORDINATES

When you set this modal you output less code. For example when you make a move in X direction only, no Y coordinate is output.

(11) 1.G CODES:G01, G02, G03
2.G CODES: MODAL (G codes are output only when you switch from one G code to another)
3.NO G CODES (For example for BANDIT)
4.TAB SEQUENTIAL

Note: BANDIT users you must also set modal (12) to RAPID CODE = /Xxxxxxx /Yxxxxxx

Note: TAB SEQUENTIAL users must also set Modal (10) to OUTPUT ALL COORDINATES

(12) NO SLASH FOR RAPID MOVE
RAPID CODE = /Xxxxxxx /Yxxxxxx

Note: BANDIT users the slash is for your Rapid moves. You must also set Modal(11) to NO G CODES

Press the N key to continue to the Next page on your screen

This page shows the following:

(Use the Up - Down Arrow keys on your keyboard to move the Modal that you want to change.
Use the Left - Right Arrow keys on your keyboard to tell the program that you want to change that Modal. Next enter the new value.)

START BLOCK NUMBERING WITH # : 5
INCREMENT BLOCK NUMBERING WITH : 5

Note: You only need to set these when you have set Modal (5) to AUTOMATIC LINE NUMBERS (see two pages back). If you do not want to display Line numbers ignore this.

**** PARAMETERS *****

YOU MUST USE A PERIOD(.) IN YOUR VALUE!

DEFAULT CLEARANCE HEIGHT: Z = .50000
DEFAULT CUTTING DEPTH: Z = -1.25000

Note: Whenever you go into the Milling Menu in BOBCAD these are the values that will show up on the screen.

NEXT: Press the N key to continue to the Next page on your screen

N.C.CONFIGURATION continued

The following page appears on your screen:

(Use the Up - Down Arrow keys on your keyboard to move to the Line that you want to change.
Use the Left - Right Arrow keys on your keyboard to tell the program that you want to change that Line. Next enter the Code.)

**** FIRST FEW LINES ****
1.
2.
3.
4.
5.
6.

**** LAST FEW LINES ****
1.
2.
3.
4.
5.
6.

Whenever you are making a new program you probably want each program to start with the same settings. By setting the lines here these settings are automatically inserted at the start of each program. For example:

FIRST FEW LINES
1. %
2. G90 X0 Y0

LAST FEW LINES
1. M02
2. %

Note: If you do use automatic block numbering you generally do not want a block number in front of a % (as shown in Line 1 in the above example) In that case start Block numbering at Line 2

NEXT: Press the **N** key to continue to the Next page on your screen.

With **BOBCAD** you can create your own **MILLING** menus. The Codes that you enter here will appear in the menus as a function. When you select that function the code is inserted as a block or at the end of a block in the N.C. program.

Use the **Up - Down Arrow** keys on your keyboard to move to the Line that you want to change.

Use the **Left - Right Arrow** keys on your keyboard to tell the program that you want to change that Line. Next enter the Code.

The following page appears on your screen:

```
****** ENTER CUSTOM BLOCK ******
1.YOUR CODE
2.YOUR CODE
3.YOUR CODE
4.YOUR CODE
5.YOUR CODE
6.YOUR CODE
7.YOUR CODE
8.YOUR CODE
9.YOUR CODE
10.YOUR CODE
****** ENTER TOOL CHANGE CODES ******
1.YOUR CODE
2.YOUR CODE
3.YOUR CODE
4.YOUR CODE
5.YOUR CODE
6.YOUR CODE
7.YOUR CODE
8.YOUR CODE
9.YOUR CODE
10.YOUR CODE
```

Note: To BOBCAD the CUSTOMER CODES and TOOL CHANGE CODES are actually the same. You can put CUSTOM CODES in the TOOL CHANGE CODES if you need more than 10 codes.

NEXT: Press the **N** key to continue to the Next page on your screen.

The following codes are added to the end of a BLOCK. Otherwise they are similar to the previous page. You can use these for example for FEED RATES. When you have entered a code here (for example F50) these will appear in the MILLING MENU in BOBCAD.

*****ENTER CODE TO BE ADDED TO END OF BLOCK**********
1. F50 (for example)
2. F1000 (for example)
3. YOUR CODE
4. YOUR CODE
5. YOUR CODE
6. YOUR CODE
7. YOUR CODE
8. YOUR CODE
9. YOUR CODE
10. YOUR CODE

The following are the names of text files. When you pick one of these names from the menu while working in **BOBCAD**, the text file with that name will be added to the N.C. program.

CUSTOM and TOOL CHANGE let you set only 1 line at a time. With MACRO you can insert as many lines as you want.

To make such a text file you must use a text editor.

************MACROS********************
1. Your Macro File
2. Macro 2
3. Macro 3
4. Macro 4
5. Macro 5
6. Macro 6
7. Macro 7
8. Macro 8
9. Macro 9
10. Macro 10

Next press the **S** key to **SAVE** the new configuration for your controller.

<u>RS-232 COMMUNICATIONS</u>

Once you have made the NC program with BOBCAD you now need to send it to your CNC Machine. There are 3 ways you can do this:

1. The way this has been done in the past was through a paper tape puncher and there are still a lot of them around. (Actually this is why BOBCAD'S NC programs end with '.TAP', the abbreviation for tape). If you use a paper tape puncher, you need ports which are also called RS-232 ports. Look in the section below on RS-232 ports and communication on how to do this.

2. The second way you can get the NC program from your computer to your CNC machine is by putting the NC-program on a floppy disk and then sticking the floppy disk into the disk drive of your CNC controller. This is not very common in the USA but if your CNC machine has a disk drive or a GRECO box then this could be an option. GRECO boxes are generally DOS compatible but in most cases the disk drives in CNC controllers are not DOS compatible! This means they cannot read a floppy disk from an IBM compatible computer. Ask the manufacturer of your CNC machine if the disk drive in your CNC controller is DOS compatible.

3. The newer way to connect the IBM compatible computer to CNC machines is through an RS-232 cable. An RS-232 cable can be bought from your local CNC specialist or we can get one for you. It does require that your CNC controller has an RS-232 port and board inside.

<u>RS-232 PORTS:</u>
On the back of your IBM compatible computer you will find several ports. Generally there are 2 COM ports and 1 printer port. COM ports are <u>male</u> and are 25 or 9 pins. <u>COM ports are also called RS-232 ports or serial ports, there is no difference.</u> The printer port is 25 pin but is <u>female</u>. The printer port is also called the parallel port. It is used for the printer. If you have only 1 COM port you will need to get a switch box or install an additional COM port because you will need one for your mouse and one for your RS-232 cable.
The mouse goes on COM port 1, the cable coming from your CNC controller goes on COM port 2.

RS-232 COMMUNICATIONS

With **BOBCAD** you received a communications floppy disk. The communications program is called **GEOTALK.** If GEOTALK has not yet been loaded just copy the entire floppy disk to the directory in which you have BOBCAD.

> **Note: If you already use a communications package to send NC programs to your CNC machine it is not necessary to use GEOTALK. You can use your communications package just as well. BOBCAD NC programs are ASCII files that can be send with almost any communications program (for example PROCOMM or BEACHCAM).**

HOW TO USE GEOTALK

After you have made an NC program you must exit BOBCAD and then type **GEOTALK** to start the communications software.
Next select function **6. SEND OUT N.C. TAPE FILE.**
Next type in the name of the NC program or hit the space bar on your keyboard to see a list of the NC programs. You can select an NC program from this list.
Next select the BAUD rate to match with the BAUD rate of your NC controller.
Next set your CNC machine ready to receive the NC program.
Next hit any key on your computer to send the NC program to your CNC machine.
Result: You will see the NC program appear on your screen as it is being sent to your CNC machine.

To exit **GEOTALK** move the cursor to function **8. EXIT GEOPROMPT** in the **MAIN MENU.**

SETTING UP YOUR CNC MACHINE FOR RS-232 COMMUNICATIONS.

When you send an NC program from the computer to your CNC machine the CNC machine must be set to receive the NC program. This is generally different for each different CNC machine. You must consult the manual or the service dealer for your CNC machine. **GEOTALK** communicates as follows:

1 Stop bit
Even parity
7 Bit word length

MENUS

Section 5

This section explains each function in the menus. They appear in the order as they are shown in the MAIN MENU.

MAIN MENU (M)

This is the **MAIN MENU** from which all functions can be reached. You can return to the **MAIN MENU** by pressing the Escape key a number of times or the right button on your mouse. You can also type the key "M" on your keyboard or select the **MAIN MENU** function at the bottom of each menu.

QUICK KEYS:
If the first letter in the function is shown with a different color in the menu on the screen then you can reach that function directly from any menu when you press that letter. **For example, when you press the E key in any menu you will go straight to the EDIT MENU.**

Type '**H**' for Help. See the '**QUICK KEYS**' section for other useful keys.

FUNCTION

1. EDIT

1) Delete Entities
2) Blank and Unblank (controls the display of Entities)
3) Trim (shorten or lengthen Lines and Arcs)
4) Break a Line or Arc into two or more parts.

2. POINT

Draw a Point free hand, by coordinates, relative to existing Entities, etc.

3. LINE

Draw a Line free hand, by coordinates, relative to existing Entities, etc.

4. ARC

Draw an Arc or Circle free hand, by coordinates, relative to other Entities, etc.

5. OTHER CURVES

Draw Rectangles, Offset Curves, Splines, Hole Patterns, Gears, Pocket and roughing, Circle clean out, Lathe cycles, Machine text.

6. COPY/MOVE

Copy, Move, Rotate, Scale, Drag, Mirror.

7. ZOOM

Set viewing area by Windowing, Scaling, or Shifting. Return to the previous zoom setting or view all.

8. LAYER

Each Entity upon creation gets the current layer number which is always shown in the lower left corner of the screen. This is handy for Blanking and Unblanking Entities.

9. VERIFY DATA

Shows values of Entities, Distances and angles between Entities, Area, and Circumference.

10. DIMENSIONS & TEXT

11. COLOR/STYLE

Set or change colors of Entities, Screen color, Menu color, Hi-lite color, Picking color, Line styles.

12. UTILITIES

Rename the drawing, file the drawing, mix in other drawings, interface to DXF (AUTOCAD(tm)), custom functions such as Standard drawings, switch X-Y axis on or off, grid, read in an NC program and turn it into a drawing.

13. NC..CAM

Milling Main Menu, Modals Menu, Built-in NC Editor, ToolUp/Tool Down Menu.

14. SAVE

Do an Instant Save of a drawing (the drawing is saved on your disk under the current name of the drawing and the extension .DGN is added).

15. QUIT

Quit and Save, or Quit without Saving.

RECOMMENDATION:

With many functions, such as picking Entities or Zooming, the speed with which BOBCAD can perform a task is affected to some degree by the number of Entities visible on the screen. If you find that a function takes a long time to pick an Entity on the screen, just reduce the number of Entities visible on the screen through the liberal use of Layers and Blanking.

1. EDIT MENU

Press the E key on your keyboard to go directly to this menu from any other menu.

1. **DELETE**
   Pick the Entitites that you want to delete. With the back-space key you can restore an Entity that was deleted with function **SINGLE DELETE.**

2. **DELETE LAST**
   Delete the last Entity that you have drawn.

3. **BLANK**
   Pick the Entities that you want to be invisible.

4. **UNBLANK**
   Pick the Entities that you want to make visible again.

5. **TRIM** (see next page)

6. **BREAK**
   Break a Line into 2 or more Lines. Break an Arc into two or more Arcs.
   1. SCREEN
      With the mouse indicate the place where you want the Line or Arc to be broken. Result: You will have 2 Lines or Arcs.
   2. POINT
      Break a Line or Arc near a Point.
   3. ANGLE
      Break an Arc at an angle.
   4. BREAK MANY
      The Entities that you pick will be broken at all the intersections.
   5. DIVIDE
      Break an Entity into N number of smaller Entities.

5. TRIM MENU continued from EDIT MENU section.

Press the T key on you keyboard to go directly to this menu from any other menu.

When you trim a Line or Arc to make it shorter you must pick the part of the Line or Arc that you want to keep! **ALWAYS PICK THAT PART OF THE ENTITY THAT YOU WANT TO KEEP!**

If you make a mistake press the **back-space** key on your keyboard while you are still in that function.

1. **SCREEN**
   With the cursor mark a position on the screen to which you want to shorten or lengthen an Entity.

2. **SINGLE**
   First pick the Entity that you want to lengthen or shorten. Next pick the entity you want to trim to. If you make a mistake press the backspace key immediately.

3. **DOUBLE**
   Pick two Entities. They will be trimmed to each other . If you make a mistake press the backspace key immediately.

4. **MODAL**
   Same as function **2.SINGLE**, but first you pick the trimmer Entity, for example a Line. Next you can keep picking the Entities to be trimmed without having to pick a new trimmer Entity. If you make a mistake press the **BACKSPACE** key immediately.

5. **STRING**
   Keep picking Lines and Arcs. They will be trimmed to form a continuous string.

Note: If the Entity you want to trim does not actually go through the other Entity it will still be trimmed to where the intersection would be.

2. POINT MENU

Press the P key on your keyboard to go directly to this menu from any other menu.

1. **SKETCH**
   Move the cursor to a position on the screen where you would like to draw a point (also called screen or free hand) and click the left button on your mouse.

2. **COORDINATES**
   Draw a point by typing in its X and Y coordinates. When in 3-D mode you also type in the Z coordinate.

3. **INCREMENTAL**
   With the cursor pick an existing Point on the screen. Next type in an X and Y distance. The new Point is drawn this distance from the Point you picked.

   Note: *If you do not type in a distance but just hit the Enter key or the left button on your mouse, BOBCAD will accept the X and Y distances typed in previously.*

4. **END**
   With the cursor pick one of the ends of a Line or Arc on the screen. A new Point is drawn at the end of the Line or Arc.

5. **INTERSECT**
   With the cursor pick two Entities on the screen. A new Point is drawn where the two Entities intersect.

   Note: *The two Entities do not actually have to intersect. A new Point will be drawn where the two Entities would intersect had they been made longer.*

6. **ARCCENTER**
   With the cursor pick an Arc or Circle on the screen. A new Point is drawn at the center of the Arc.

7. **PERPENDICULAR**
   With the cursor pick a Point on the screen. Next pick a Line or Arc. A new Point is projected onto the Line or Arc you picked, perpendicular from the old Point.

8. ON ENTITY

With the cursor pick a Line or Arc from the screen. When picking a Line you must type in a fraction of its length from the end where you picked the Line. A new Point is drawn on the Line. For example, if want a Point in the middle of the Line type in 0.5 . When you have picked an Arc or Circle type in an angle (counter-clockwise from the X-axis in degrees).

3. LINE MENU

Press the L key on your keyboard to go directly to this menu from any other menu.

1. SKETCH

   Move the cursor to a position on the screen where you want the start of the new Line to be and press the left button on your mouse. Next move the cursor to a position on the screen where you want the end of the new Line to be and press the left button on your mouse. Next click the right button on your mouse and a new Line is drawn.

   Note: This function is also called free-hand mode and is not accurate.

   OPTIONAL SNAP: The software detects whether the Line you have drawn is more horizontal or more vertical. If the Line is more horizontal you can type in any Y coordinate. The Line will then become perfectly horizontal. If the Line you have drawn is more vertical you can type in any X coordinate. The Line will then become vertical through that X-coordinate. To keep the Line at an angle hit the right button on your mouse or the Esc key.

2. COORDINATES

   Draw a Line by typing in the start coordinates and the end coordinates of the Line. When 3-D mode is turned on you will be asked for the Z coordinate.

3. JOIN

   With the cursor pick two Points or the ends of a Line or Arc. A new Line is drawn that joins the two Entities you picked.

4. TANGENT

   With the cursor pick two Arcs on the screen or one Arc and one Point. A Line is drawn tangent to the Arcs on the sides where you picked them.

   Note: If you want to draw a Line that is tangent to only one Arc at a specific angle, you must use the 6. ANGLE/POLAR function.

5. PARALLEL

With the cursor pick a Line on the screen. Next move the cursor to the side where you want the new Line to be and pick that side. Next type in a distance and a new Line is drawn this distance from the Line you picked.

Note: *If you do not type in a distance but just hit the enter key or the left button on your mouse, BOBCAD will use the distance that is shown in the menu. With the OFFSET: SINGLE function in the OTHER CURVES menu you can do the same thing faster and also make parallel Arcs.*

6. ANGLE/POLAR

With the cursor pick a Point, Line or Arc on the screen as the start of the Line. Next type in the length and the angle (counter-clockwise in degrees). A new Line is drawn with this length.

Result: *The angle of the new Line is measured from the X-axis. In addition, the start of the new Line is from the end closest to where you picked the old Line. If you picked an Arc, the new line is tangent to the Arc and the angle of the new Line is from the X-axis. The Line is tangent on the side of the Arc where you picked it.*

7. CHAMFER

With the cursor pick two Lines on the screen. Next type in an angle and a distance. A new Line is drawn as a chamfer and the two Lines you picked are trimmed (just like a Fillet). The angle of the new Line is to the first Line you picked and the distance is along the first Line you picked.

8. CONTINUOUS (Also called String)

First you will be asked to create the start of the first Line. Next create the end of the first Line. The end of this new Line is now the beginning of the next Line. More lines can be drawn from this point by simply creating end points.

4. ARC MENU

Press the A key on your keyboard to go directly to this menu from any other menu.

NOTE: In the CAD, Arcs and Circles are drawn counter clockwise around the center of the Arc. 0 degrees is in the positive X-axis direction, 90 degrees is in the positive Y-axis direction, etc. This rule does not affect the CAM or NC output. The CAD does not make a difference between an Arc and a Circle. In most cases the Circle starts at 0 degrees and then ends at 360 degrees. The start and end point of a Circle are the same.

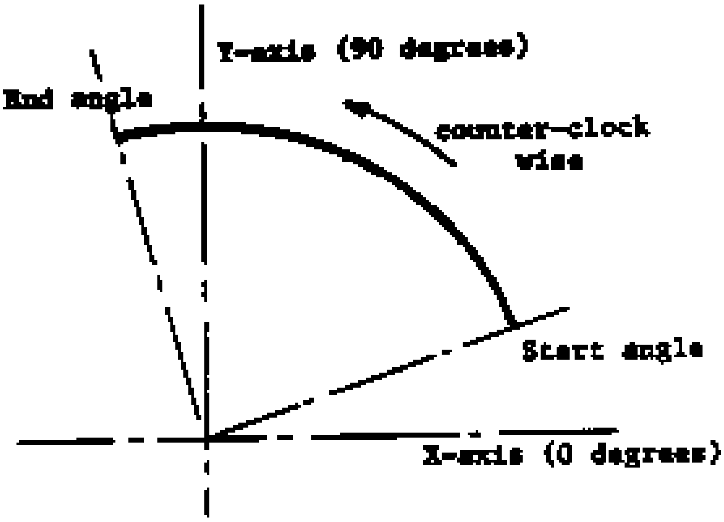


Figure shows how Arcs and Circles are stored in BOBCAD.

1. SKETCH CENTER
   Type in the radius, the start angle, and the end angle. Next move the Arc or Circle to the position on the screen where you want the center of the new Arc or Circle to be.

2. COORDINATE CENTER
   Draw an Arc or Circle by typing in the X and Y coordinates of the center, the radius, and the start and end angles.

3. POINT CENTER
   Type in the radius, the start angle, and the end angle of the Arc or Circle. Next with the cursor pick a Point on the screen where you want the center of the new Arc or Circle to be.

4. FILLET

With the cursor pick two Entities. Next enter the radius. A new Arc is drawn tangent to the two Entities (similar to a round corner). The Entities are automatically trimmed. You can turn trimming off by hitting the **F1** key before you pick the first entity (see the prompt at the bottom of the screen).

Note 1: *Arcs are drawn counter-clockwise. The first Entity that you pick will become the start of the new Arc. The second Entitiy that you pick will become the end of the new Arc. However, when you make a Fillet between two Lines you can pick the Lines in any order.*

Note 2: *If you do not type in a radius but just hit the Enter key or the left button on your mouse, BOBCAD will use the default radius.*

5. 3 ENTITIES

With the cursor pick 3 Entities (Points, Lines or Arcs in any combination). An Arc will be drawn tangent to the three Entities you picked.

Note: *The first Entity you picked will be the start of the new Arc. The last Entity you picked will be the end of the new Arc. (There is also an exercise in the CAD section of this manual called **ARC TOUCHING 3 ENTITIES** that shows you how this function works).*

6. CHAIN

With the cursor select a shape that consists of Lines and Arcs. Next type in a radius for left turns and a radius for right turns. Fillets will be drawn in each sharp corner. This function works only for sharp corners between Lines and does not put a Fillet between a Line and an Arc or between Arcs.

Note: *This function is important for older types of controllers that need a Fillet in each sharp corner.*

7. ELLIPSE

Draw an Ellipse. In the CAD the Ellipse is drawn as a number of short Arcs because most CNC machines have no Ellipse function.

5. OTHER CURVES

Press the O key on your keyboard to go directly to this menu from any other menu.

1. RECTANGLE

This function asks you to input the two opposite corners of the Rectangle. Next enter the corner radius or type in 0.

2. OFFSET (Cutter Compensation)

1. SINGLE Enter an offset distance and with the cursor, pick a Line or an Arc. Then pick the side of the Line or Arc on which you want the offset curve to appear.

2. CHAIN (Machine curve) This function is very usefull for programming tool center. It creates an offset curve that also puts a radius around sharp corners and checks for tool interference. (See also the advanced drawing exercise: OFFSET CURVE.) After you have made an Offset curve you can repeat with Offset curves for spiral roughing of a pocket. Use the CONNECT function shown below to connect the Offset Curves.

3. CONNECT. Use this function to connect Offset curves (created with the above function) when you want to draw a roughing profile. (See also the advanced drawing exercise OFFSET CURVE CLIMB POCKET).

3. SPLINE

With this function you can draw a smooth curve through a set of Points. The Spline is drawn as small Lines which can then be machined as any Line.

Note: When you pick the Points on the screen you must pick them from left to right or from bottom to top.

4. HOLE PATTERN

Draw bolt holes in a grid pattern or circular pattern.

5. **POCKET** (Roughing) See also the exercise called ROUGHING A POCKET)
This function draws a roughing path inside any closed shape. The shape can include many islands. The roughing path is drawn as Lines and Arcs. You can turn these Lines into machine code in the CAM menu.

TO OPERATE:
First select the shape.
Next: Enter the tool diameter.
Enter the Stepover distance (about half the tool diameter).
Enter the Stock distance (amount of material to leave on the shape).
Result: An offset is drawn.
Next: Pick any islands or hit the right button on the mouse.
Result: The roughing path is drawn on layers 1000 to 1002.
There may be dashed Lines where the tool crosses an island. When the CAM portion sees a dashed Line the tool automatically picks up and rapids over.

6. **CLEAN CIRCLE**
Pick a Circle and BOBCAD will draw a spiral roughing path to clean out the circle. The roughing path can then be machined in the NC..CAM menu.

7. **TEXT PATH**
First draw the Text in the DIMENSION MENU (You can set the size and angle with the ATTRIBUTES function in the DIMENSIONS MENU). Next select the TEXT-PATH function to turn the characters into short Lines. The Lines can then be machined in the NC...CAM MENU. In the CAM MENU always start cutting the text at the lower left corner of the first character. Then use the AUTO function to cut all the way to the end. The dashed Lines become Rapid moves in clearance.

8. **GEAR**
INVOLUTE GEAR. (See also the INVOLUTE GEAR section at the end of this manual). The Involute gear is drawn with small Arc segments which can then be machined in the NC...CAM menus (G02, G03).

6. COPY/MOVE MENU

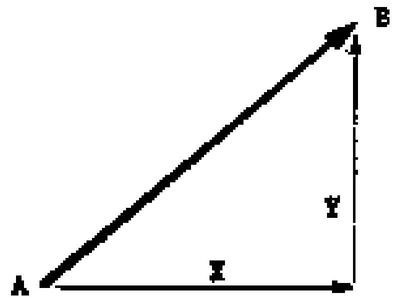
COPY, MOVE, ROTATE MIRROR, SCALE, DRAG

(See exercises for examples)

***Note:** With these functions you can Copy, Move, Rotate, Mirror, Scale and Drag Entities. In order to scale Entities use the TRANSLATE or ROTATE functions and change the Scale before selecting the vector mode.*

WHAT IS A VECTOR?

A vector is like an arrow on the screen. The arrow tells where you want the Entities to be moved to (for example from Point A to Point B). If you select function SCREEN or function POINTS from the VECTOR MODE MENU, BOBCAD asks you for Points A and B. If you select function INCREMENTAL X, Y from the VECTOR MODE MENU the Entities will be moved over distance X,Y.



COPY FUNCTIONS

1. TRANSLATE
2. ROTATE
3. MIRROR

1. TRANSLATE

With this function you can make copies of Points, Lines, Arcs and Dimensions at new locations. There are 4 ways to move the Entities.

1.SKETCH: First pick the Entities to be copied. Next with the cursor indicate a 'from' position on the screen followed by the 'to' position.

2. BY POINTS: First pick the Entities to be copied. Next with the cursor pick the 'from' Point followed by the 'to' Point.

3. DRAG: First pick the Entities to be copied. Next move the mouse to drag the copy around. When the copy is where you want it click the left button on your mouse. To get out of this function click the right button on your mouse.

4. INCREMENTAL X-Y: First pick the Entities. Then type in the distance to be moved.

Note: If you want the copies to be smaller or larger you must set function 5.SCALE in the VECTOR MODE MENU.

2. ROTATE

With this function you can make copies of Entities and rotate them. There are two ways to rotate the Entities.

1.SCREEN: First pick the Entities to be rotated. Next indicate the location to rotate around.

2. POINT: First pick the Entities to be rotated, Next pick the Point to rotate around.

Note: If you want the copies to be smaller or larger you must select function 3.SCALE before selecting function 1 or 2 in the VECTOR MODE MENU.

3. MIRROR

First pick the Line in which you want to mirror the Entities. Next pick the Entities that you want to mirror.

<u>MOVE FUNCTIONS</u>

6.TRANSLATE
7.ROTATE
8.MIRROR

6. TRANSLATE

With this function you can move Points, Lines, Arcs and Dimensions to a new location. There are 4 ways to move the Entities.

1.SKETCH: First pick the Entities to be moved. Next with the cursor indicate a 'from' position on the screen followed by the 'to' position.

2. BY POINTS: First pick the Entities to be moved. Next with the cursor pick the 'from' Point followed by the 'to' Point.

3. DRAG: First pick the Entities to be moved. Next move the mouse to drag the part around. When the part is where you want it click the left button on your mouse. To get out of this function click the right button on your mouse.

4. INCREMENTAL X-Y: First pick the Entities. Then type in the distance to be moved.

Note: If you want to reduce or enlarge the part you must set function 5.SCALE in the VECTOR MODE MENU.

7. ROTATE

With this function you can rotate Entities. There are two ways to rotate the Entities.

1.SCREEN: First pick the Entities to be rotated. Next indicate the location to rotate around.

2. POINT: First pick the Entities to be rotated, Next pick the Point to rotate around.

Note: *If you want the part to be smaller or larger you must select function 3.SCALE before selecting function 1 or 2 in the VECTOR MODE MENU.*

8. MIRROR

First pick the Line in which you want to mirror the Entities. Next pick the Entities that you want to mirror.

7. ZOOM MENU

Press the Z key on your keyboard to go directly to this menu from any other menu. When you are done you automatically return to the function you where In by pressing the right button on your mouse.

Note 1 : The ZOOM functions do not change the size of the Entities and parts. The Zoom function works just like the Zoom lens on a camera - it lets you look at things close up or far away. If you want to make a part bigger (not just look bigger) you must use the COPY/MOVE functions.

Note 2 : The ZOOM MENU can also be called when a cursor is on the screen. After you have zoomed hit the right button on your mouse to return to where you were.

Note 3 : You can also redraw the screen by pressing the **R** key.

1. **WINDOW**
   Set viewing area by creating a rectangular region with the cursor.

2. **NEW SCALE**
   Set viewing area by typing in the scale. Next with the cursor indicate the location that will become the new center of the screen, or hit the '[' key (see top line on screen) and type in the X and Y coordinates of the new lower left corner.

3. **VIEW ALL**
   A viewing area is created with all Entities displayed (except for blanked Entities).

4. **SHIFT**
   Shift the current view over. Move the cross-hairs of the mouse to where you want the center of the view to be.

5. **PREVIOUS**
   Returns to the last view.

6. **REDUCE**
   Makes the view smaller by doubling the scale.

7. 3-D VIEWS

1. TOP VIEW : X - Y PLANE

2. FRONT VIEW : X - Z PLANE

3. SIDE VIEW : Y - Z PLANE

4. DEFAULT 3-D VIEW

5. ISOMETRIC VIEW

6. ROTATE MOD. : Rotate the view around the axis displayed in the upper right hand corner of your screen

7. ROTATE SCREEN AXIS: Rotate the view around the X axis, the Y axis and the Z axis of the screen.

8. ORIENT :

   Pick the new origin point.
   Next pick another point. The line from the origin point to this point forms the new X-axis.
   Next pick a third point. This point defines the new X-Y plane.

****

Note: After a Zoom, the X-Y coordinates of the lower left corner are displayed in the lower left corner.

****

8. LAYER MENU

When you draw an Entity, it automatically gets the layer number that is shown in the lower left corner of your screen. This is handy for such functions as Blanking, Unblanking and Deleting. For example, when you Blank layer 10 to 11 all Entities that have the layer number 10 or 11 will be blanked.

Note: *Function DRAW PATH in the UTILITIES MENU puts the shape automatically on layer 98 and 99. Function POCKET in the OTHER CURVES MENU puts the roughing path automatically on layers 1000-1002.*

1. NEW

Change the current layer number (shown in the lower left corner of your screen). From this point on, all new Entities drawn will have the new layer number. You can use from layer 0 to 999.

2. SHOW ENTITY

Pick an Entity to see which layer it is on.

3. MODIFY ENTITY

Pick Entities and put them on a different layer.

4. BLANK

1. Pick an Entity. All Entities on the same layer as the Entitiy you picked will be blanked.

2. Enter the layer # to be blanked.

Note: You can also use the BLANK functions in the EDIT MENU.

5. UNBLANK

Unblank a layer.

Note: You can also UNBLANK in the EDIT MENU.

9. VERIFY DATA

Press the V key on your keyboard to go directly to this menu from any other menu.

1. SINGLE

Pick an Entity with the cursor to display its values. In addition to the data base values, additional values will be displayed such as length and angle of a Line.

2. 2 ENTITIES

With the cursor pick two Entities. The distance or the smallest angle between them will be calculated. When the values are displayed you can continue to pick two new Entities with the cursor.

3. AREA / CIRCUMFERENCE

With the cursor pick a shape. The area and the circumference of the shape will be calculated.

10. DIMENSIONS AND TEXT MENU

Press the D key on your keyboard to get to this menu from any other menu.

If you press the '[' key (shown in the prompt line on your screen) you can switch from ACTUAL to ENTERING dimensions.

The size of the Dimensions can be set in the ATTRIBUTES function.

See additional notes at bottom of these pages.

1. HORIZONTAL

   Draw a horizontal dimension between two Entities. Pick the first Entity. Next pick the second Entity and with the cursor pick the position where you want the value to be.

2. VERTICAL

   Draw a vertical Dimension between two Entities. Pick the fist Entity Next pick the second Entity and with the cursor pick the position where you want the value to be.

3. OTHER

   Draw a Dimension between two Entities. The direction of the Dimension is determined by:

   1) The Line if only one of the Entities is a Line.
   2) By the first Line you picked if both Entities are Lines.
   3) By the direction of the shortest distance in all other cases.

4. ANGLE

   Draw a Dimension showing the angle between two Lines. Pick the first Line. Pick the second Line counter-lockwise from the first Line. Next pick the position where you want the value to be. If you press the backspace key immediately after the Dimension is drawn you can move the value to a different location.

5. RADIUS

   Draws a Dimension showing the radius of an Arc or Circle. Pick the Arc. Next pick the position where you want the Radius to be. If you press the back-space key immediately after the Radius is drawn, you can move the value to a different location.

6. TEXT

Place text on the screen. To set the size and angle of the text use function 8. **ATTRIBUTES.** To turn the text into Lines so they can be machined use function **TEXT PATH** in the **OTHER CURVES MENU.**

7. ATTRIBUTES:

1. **SIZE** . Set size of DIMENSION and TEXT (in Millimeters or Inches) Default = 2 Millimeters / 0.125 Inches

2. ANGLE. Set the angle of Text before it is drawn.

3. DECIMALS. Increase or decrease the number of digits that are displayed behind the period in a Dimension. The range is from 0 to 4 decimal places.

4. SCALE FACTOR. This function lets the Dimensions come out to a different scale. For example, if you enlarge a part to twice its real scale so that it comes out bigger on a drawing, then when dimensioning it you can set the SCALE FACTOR to 1/2 and your Dimension will come out correct.

8. HATCH

Shapes can be cross-hatched. Specify the distance between the cross-hatch Lines and the Angle. The cross-hatch is built up of individual Lines and is put on layer 600.

9. MOVE / ALIGN

1. MOVE A DIMENSION.

2. ALIGN HOR.: Pick a Horizontal Dimension. Next pick other Horizontal Dimensions. These will now be level with the first Dimension that you picked.

3. ALIGN VERT. : Pick a Vertical Dimension. Next pick other Vertical Dimensions. These will be aligned with the first Dimension that you picked.

11. COLOR MENU

Note: To change the default colors when BOBCAD starts up use program CADCFG from DOS.

1. ACTIVE COLOR
Sets the color with which new Entities will be drawn.

2. MENU COLOR
Sets the color of the menu, cursor, X-Y axis and other information on your screen. (Press the R on your keyboard to do a Repaint and you will see the X-Y axis update its color.

3. HI-LITE
Set the color of the Hi-lited bar in the menu.

4. SCREEN COLOR
Set the color of the screen.

5. PICKING COLOR
Set the color shown when picking an Entity.

6. MODIFY COLOR
Change the color of existing Entitles.

7. DRAW STYLE
Choose between Solid and Dashed Line-styles for drawing Entitles.

***Note:** When the CAM portion of BOBCAD goes over a dashed Line it will automatically produce a rapid move. Knowing this feature can be very usefull*

8. MODIFY STYLE
Change the Line-style of existing Entities (Solid or Dashed)

***Note:** When the CAM portion of BOBCAD goes over a dashed Line it will automatically produce a rapid move. Knowing this feature can be very usefull*

12. UTILITIES MENU

Press the U key on you keyboard to go directly to this menu form any other menu.

1. CUSTOM

1. X-Y AXIS display. Select this function and the X-Y Axis will be displayed or turned off.

2. STANDARD DRAWING When you make a drawing that you want to use over and over, you should save it with one of the names STD1 to STD8. Next time you want to use or merge that drawing, select it from this menu and the drawing is automatically merged into the current drawing.
This is also usefull Bills of materials, company logos etc. that you want to add to the drawing once you are done.

3. GRID When you turn on the grid all Entities that are drawn with the sketch functions (for example POINT: 1.SKETCH) will snap to the nearest grid location.

4. DIGITIZE
   1. DEFAULT
   2. 1 TO 1

When you want to use a digitizer you need to configure BOBCAD. Type CADCFG while in the BOBCAD directory. Next set TABLET/MOUSE/NONE to a digitizing tablet. If your tablet is 18x12 or larger you must use the 18x12 tablet setting.
Tablets must be initalized to high resolution when you turn on your computer. (For example when running K_CONFIG for a KURTA tablet)
Inside the CAD you can run digitizers in 2 modes:

1. DEFAULT
   Set to default mode to allow easy movement of the cursor over a section of the tablet.
2. 1 TO 1
   Use this mode for accurate 1 to 1 digitizing of a drawing.

2. RENAME DRAWING

Select this function to change the name of your drawing. (The current name of your drawing is shown in the prompt line). If you do not enter a name but just hit the Enter key the current name is not changed.

3. READ DXF :

With this function BOBCAD can read in a drawing made on other CAD systems. The drawing that you want to read in must be in DXF format, which is originally an AUTOCAD format. After you select this function, type in the full name of the DXF file. The other way around to turn a BOBCAD drawing into a DXF file use the separate program DGN2DXF (while you are in DOS).

Note1: *BOBCAD reads in Points, Lines, Arcs and Circles from DXF files. Dimensions cannot be read in in a useful way. Since BOBCAD has a limit of 11,000 Entities it is important to tell the person who is giving you the DXF file to only give you the Entities you need.*

Note2: *Be aware that a DXF drawing that looks good on the screen or on paper may not be good enough for machining. After you have loaded in a DXF file verify the Entities to make sure the drawing was created accurately.*

4. FILE

1.SAVE (You can press the S key on your keyboard to do an instant save from any menu). The drawing is saved automatically under the name with which you started. (The extension .DGN is automatically added to the end of the name). (By the way, if you forgot the name and want to know it, use function 4.RENAME in the UTILITIES MENU to find out).

2.MERGE Select this function to add an existing drawing (previously saved) to the current one. You can only merge drawings that have been saved in the BOBCAD format. Such drawings have the extension .DGN. If you want to merge a DXF file you must use the function 3.READ DXF in the UTILITIES MENU.

3.ASCII SAVE When you select this function the current drawing is saved in ASCII fromat. BOBCAD saves Points, Lines, Arcs, etc. If you want to be able to do something with these files, like develop other features, call BOBCAD-CAM for further information and help.

4.ASCII MERGE Merge an existing drawing that was saved in ASCII format.

5. THREE-D

(see next page)

6. DRAW PATH

This function reads in an N.C. program (G code file) and turns it into a drawing. This function is limited to recognizing G00, G01, G02, G03, G81, G90 and G91 commands. These commands define geometry.
G00 codes are turned into dashed Lines.
G01 codes are turned into Lines.
G02 and G03 codes are turned into Arcs.
The dashed Lines are drawn on layer 98. All other Entities are drawn on layer 99 for your conveniance.

NOTE:
The system checks the first 10 lines of the NC program you are reading in and looks for a G90 (absolute programming) or G91 (incremental programming). If neither is present you will be asked if your NC program is absolute or incremental. The function also checks to see if you have decimal points in your program, otherwise you will be asked to define the accuracy.
Once you have entered the full name of the NC program (including the extension, for example: .TAP) you can step through the program by repeatedly hitting the 1 key (for 1 step at a time) or the 2 key (for 2 steps at a time) etc.

3-D MAIN MENU

1. TURN 3-D ON / 2-D ONLY

Select this function to switch between 2-D mode and 3-D mode. 3-D mode must be on for 3-D VIEWS to work in the ZOOM MENU. With 3-D mode on you will also be asked for the Z coordinate in many of the functions. For example in the POINT COORDINATE function and the LINE COORDINATE function.

The first time you turn 3-D on you will automatically get the Default 3-D view.

2. INCREMENTAL / ABSOLUTE

Select this function to switch between Incremental Z and absolute Z mode. This has an effect on functions 3.MOVE TO Z, 4.Z FROM POINT Y, and 6.CROSS SECTION. With this function set to ABSOLUTE the Entities will have their Z coordinate changed to the Z coordinate that you entered or supplied. With this function set to INCREMENTAL the Entities get the new Z coordinate added onto their previous Z coordinates.

3. MOVE TO Z

The Entities that you pick will be moved to the Z depth that you enter. With function 2 set to INCREMENTAL the Entities will be displaced in Z over the distance that you enter.

4. Z FROM POINT - Y

The Entities that you picked will be moved to the Z depth that is gotten from the Y coordinate of the Point that you picked. When function 2 is set to Incremental the Entities will be moved in Z over the Y coordinate of the Point that you picked.

5. 3 POINTS

6. CROSS SECTION

(For examples see the exercise called CROSS-SECTION in the CAM section of the manual)

1.FRONT VIEW: Combine a top view with a front view to create a 3-D part.

2.SIDE VIEW: Combine a top view with a side view to create a 3-D part.

7. INTERPOLATE:

To divide an Arc or a Line into small Lines. The Entities that you interpolate will be blanked and moved to layer 100.

Purpose:

1. If you want to machine a 3-D Arc that is not in the X-Y plane then interpolate the Arc into Line segments.

2. To divide an Arc or Line into small Line segments for the function RULED: SURFACE. (See function SURFACE for details) .

HOW TO INTERPOLATE

Select function INTERPOLATE

Next you see the following menu:

ENTER ACCURACY / NUMBER OF DIVISIONS
ACCURACY =
DIVISIONS =
MAX. LENGTH =

There are 3 ways to decide how to break up the Entity, 1. By setting the maximum deviation from the original shape (in case of an Arc), 2. By entering the number of divisions you want. 3. By setting the maximum length of each segment.

The system will use the value that will result in the most divisions. When interpolating a curve to be used in function SURFACE: RULED you must enter the number of DIVISIONS and leave the other settings the way they are.

8. SURFACE

Draw a machining surface between 2 curves. These curves can be made up from Lines and Arcs.

1.RULED.

This function draws a ruled surface path between two 3-D curves. The Entities of the curves must have been Interpolated previously with function 3-D MAIN MENU: INTERPOLATE. This function allows finer control than the next function 2.AUTO RULED.

How to operate:

1. Draw the 2 curves.

2. Interpolate each Entity in the curves. Make sure the total number of segments is equal between the 2 curves.

3. Use function SURFACE RULED. Make sure you pick each curve at its beginning. The Sytem draws a straight Line from the end of each Line segment that it finds.

4. These Lines can then be turned into NC code with the CAM menus.

2. AUTO

This function makes a ruled surface between any set of 3-D curves (see exercises). This operates similarly to the above function RULED but the Entities of the two curves are automatically interpolated. During interpolation the original Entities are blanked and saved on layer 100. The number of Entities has to be equal in the 2 curves. You can use the BREAK function in the EDIT MENU to make them equal.

3. FOLLOW CURVE

(see exercises: DISH and SPHERE in the CAD section of this manual and also exercise P-TRAP in the CAM section of this manual.

TO OPERATE:

1. Draw a curve consisting of Entities.
2. Draw another curve and interpolate it.
3. Select function SURFACE: FOLLOW CURVE. Pick the first curve and pick the second curve.

13. NC..CAM

The following menu appears when first entering the NC-CAM section of Bobcad:

NC FILE OPTIONS
1. NEW FILE
2. OLD FILE
3. LAST FILE

1.NEW FILE
Select when making a new NC program from the part you have drawn on the screen. Now enter any name you want to give the NC program, for example ACME1 (the name should be no longer than 8 characters). When finishing the program and exiting Bobcad, the program will be saved under the name ACME1.TAP.

To make another NC program from the same drawing hit the F1 key on your keyboard while you are in the **WIRE EDM MAIN MENU.** (A prompt on the screen will tell you when you can press the F1 key)

2.OLD FILE allows you to continue with an old NC program.

3.LAST FILE will allow you to continue with the last NC program made.

MILLING MAIN MENU

F1: FILE End current CNC program and start a new program

F2 : EDIT Edit functions (also function 6 in this menu, see description on following pages)

1. MODALS (see description on following pages)

2. DRILL
With this function, pick Points, Arcs, or Circles to be drilled. The Point must be drawn with the **POINT MENU** or using function **HOLE PATTERN** in the **OTHER CURVES MENU.** Many drill functions are supported. After selecting a drill function, hit the F1 key for region

select and the F2 key to enter drill variables. G80 turns drill mode off.

3.POINT MOVE

With this function, a linear 2-d or 3-d move to a Point or the end of a Line or Arc can be made. The move will be rapid or feed depending on function 8.

4.SINGLE

This function allows you to cut a contour making a single cut at a time. If the tool is not yet on the contour, use function **POINT MOVE** (shown above) to bring the tool to the contour. If necessary, bring the tool down with the **TOOL UP/DN** menu function. Then select function **SINGLE** . With the cursor, pick the Line or Arc that you want to cut. The tool should already be at the start of the Line or Arc. Point the arrow in the direction of the cut.
Result: The Line or Arc is cut.
Select function **SINGLE** again and the next Line or Arc in the contour gets cut automatically.

When function **SINGLE** comes to a junction in the contour or the end of a contour, you will be asked to pick the element to be cut.

When function **SINGLE** finds a dashed Line (see **COLOR** menu), then a tool up move is generated followed by a rapid move to the end of the Line followed by a tool down move.

Note: In the **TOOL UP/DN** menu, function **Z=FIXED** results in 2-D output. Toggle this function to **Z FROM CURVE** and you will get 3-D output.

5. AUTO

Read function **SINGLE** described above before trying this function. Function **AUTO** will cut an entire shape.

Use the function **POINT MOVE** to move the tool to the beginning Line or Arc. Next go to the **TOOL UP/DN** menu.
Use function **TOOL DOWN** for 2-D or 8 for 3-D.
Next select function **AUTO** and pick the first Line or Arc (The tool should already be at the beginning of the Line or Arc).
Point the arrow in the direction of the cut.
Next pick the last element on the shape that you want cut or hit the ']' key to cut the entire shape.
Result: The elements will be cut.

Note: Function **SINGLE** and **AUTO** can be used combined.

Note: When function **AUTO** meets a dashed line (see **COLOR MENU**) the tool is automatically moved up to clearance. A rapid move is then created along the dashed line followed by a feed move down into the part.

6.EDIT

While creating your CNC program, BOBCAD allows you to make changes to the CNC program with the Editor. The **EDIT MENU** also lists **TOOL CHANGES, MACROS** and other utilities. The following pages contain a section devoted to the **EDIT MENU**.

7. TOOL UP/DN

With this menu you can bring the tool up or down and set your cutting depths. The following pages contain a section devoted to this menu.

8.RAPID MODE / FEED MODE

Select this function to toggle between Rapid (G00) and Feed modes (G01). This setting affects functions **POINT MOVE, SINGLE** and **AUTO.**

9.CUTALL

This function cuts all the contours, with the exception of those in a layer that has been blanked. When this function is chosen, a rapid move is made from the origin to the closest element. The element and the connecting contour is then cut. **CUTALL** then finds the next nearest contour and cuts it. When all contours have been cut the tool returns to the origin.

Note: **RAPID MODE/FEED MODE** should be set to **FEED MODE** when using this function.

MODALS MENU

G40 CUTTER COMPENSATION OFF.

Click this function and 'G40' is added to the CNC code. G40 is a command that is recognized by machines that support cutter compensation.

G41 : TURN CUTTER COMPENSATION ON TO THE LEFT

Click this function and 'G41' is added to the cnc code. G41 is a command that is recognized by machines that support cutter compensation.

G42 : TURN CUTTER COMPENSATION ON TO THE RIGHT

Click this function and 'G42' is added to the cnc code. G42 is a command that is recognized by machines that support cutter compensation.

G90 : PROGRAM IN ABSOLUTE MODE

G90 tells the CNC controller that the code is in absolute coordinates relative to the origin. The origin can be reshifted with function NEW ZERO AT in this menu.
With program MILLCFG.EXE you can set the default value to G90 or G91. Here you can override it.

G91 : PROGRAM IN INCREMENTAL MODE

G91 tells the CNC controller that the code is in incremental coordinates. With program MILLCFG.EXE you can set the default value to G90 or G91. Here you can override it.

G92 :Make a G92 move to a selected Point.

NEW ZERO AT:

While programming in absolute mode (G90) you can change the reference point. With this function pick a Point on the screen. This Point will now become the new absolute origin from which all X and Y coordinates in your CNC code are measured.

TOOL DISPLACEMENT:

This function is for machines with more than one head (For example, overhead gantry routers with several mounted router spindles). Typically such routers are displaced with an X and Y displacement.

Built-In NC Editor

In the **MILLING MAIN MENU**, the **EDIT MENU** can be entered by pressing the F2 key (a prompt on the screen will indicate this).

EDIT MENU
EDIT BLOCK
OOPS
TOOL CHANGE
CUSTOM
ADD TO BLOCK
MACROS
NEW BLOCK #
SIMULATE

1. EDIT BLOCK

When selecting this function, a white text cursor will appear near the bottom of the screen on top of the NC program. An arrow also appears, pointing to that part of the shape which corresponds to where the cursor is in the NC program. When moving the cursor with the mouse, the arrow also moves, indicating the part of the shape being edited.

Move the text cursor to the part of the program that you want to change and click the left button on the mouse. The bright white cursor will move to the place chosen. Start typing in the changes or hit the space bar to erase what is there. Clicking the left button on the mouse will also erase text. Note that anything typed in is added where the bright white cursor is, not where the grey cursor is.

To exit the Editor hit the Esc key or the right button on the mouse.

2. OOPS

This function allows the user to delete the last line of code from the NC Program.

3. TOOL CHANGE

Select the tool you want to add to the NC code. The ten tools listed can be changed and set with the program MILLCFG. To run MILLCFG, exit the Bobcad program.

4. CUSTOM BLOCK

Select the code to be added to the NC code. The ten Custom commands listed can be changed and set with the program MILLCFG. To run MILLCFG, exit the Bobcad program.

5. ADD TO BLOCK

ADD TO BLOCK commands are added to the end of the last NC blok on the bottom of the screen. The ten ADD TO BLOCK commands can be changed and set with the program MILLCFG. To run MILLCFG, exit the Bobcad program.

6. MACROS

The MACROS list is a list of text files, created using any ASCII text editor, that contain any number of ASCII text lines. Any section of program code that requires more then one line of code can be placed into a MACRO file, such as canned cycles, entire tool change blocks, a single pass of lathe threading code, or any semi-repetitious sequence of g-code.

Where CUSTOM and TOOL CHANGE set one line of code at a time, a MACRO file can contain an unlimited number of lines. The data is static and no variables are used or calculated within the MACRO file.

The text file name can have any valid DOS extension. A .MAC extension is recommended so that the files are easier to find and do not get confused with other data-type files. The name of the file and its extension must be added to the MACRO list in the MILLCFG file (see the NC CONFIGURATION section of the manual). Then in BOBCAD, the file name appears in a list when MACRO is chosen from the EDIT MENU. When the file name is then chosen, the text in that file is added to the current NC program.

7. NEW BLOCK #

This function is used only when NO LINE NUMBERS has been set to AUTOMATIC LINE NUMBERING in the file MILLCFG (see the NC

CONFIGURATION section of the manual). This function can be used to have the NC program continue at a new block #.

8. SIMULATE

Given a tool path diamenter, SIMULATE will display a simulation of the tool diameter cutting the tool path. This allows you to see how the cutter would remove the material for the path given. SIMULATE can be used before creating the G-Code or after a G-Code file has been produced and is read in to BOBCAD using the DRAW CNC utility.

For Example, in machining a pocket in which the OFFSET command was used to offset a shape toward the center of the pocket, SIMULATE can be used to check the path of the tool and assure that all of the material is removed.

TOOL UP / TOOL DOWN MENU

1. Z = 0.100

This number refers to the clearance height, which can be modified with this function. The default setting can be adjusted with program MILLCFG.EXE.

2. TOOL UP

Select this function to move the tool up to the clearance height. The tool on the left-hand side of your screen will represent the up or down position.

3. TOOL DOWN

Select this function to move the tool down to the cutting depth shown in function 5. The tool on the left-hand side of your screen will represent the up or down position.

4. RAMP

This function moves the tool from clearance height to a Point or the end-Point of a Line or Arc for a 3-D move.

Note: in order to ramp to a contour you must toggle function Z = FIXED to Z FROM CURVE.

5. Z = -1.2500

This function controls the cutting depth of the tool. Before moving the Tool down with the **TOOL DOWN** function, the depth can be changed with this function. This depth is used for cuttting at fixed depths (also called 2.5 axis cutting). To get 3-D surfaces set function **Z=FIXED** to **Z FROM CURVE** and your cutting depth will come from the contour. (see exercises on 3-D surface machining). The default setting can be adjusted with function MILLCFG.EXE.

Z = FIXED/ Z FROM CURVE

Toggle function :

Use **Z = FIXED** when cutting a 2-D path at fixed depth. The depth in function 5 is used.

Use **Z FROM CURVE** when cutting a 3-D contour.

QUICK KEYS AND CURSOR CONTROL

Section 6

DIGITIZING TABLET

IMPORTANT KEYS

When pressing a Quick Key, you go to that menu instantly. Looking at the MAIN MENU on the screen the Quick Keys are the letters that have the same color as the Hi-Lighted bar. For example, when you are in the ARC MENU and you press the P key on the keyboard, you go instantly to the POINT MENU.

Quick Keys only work when there is a menu on the screen!

Note: When the cursor is on the screen, the Keys P, L, A, and D have a different function (see next page under the title IMPORTANT KEYS WHEN THE CURSOR IS ON THE SCREEN).

QUIK KEYS
IMPORTANT KEYS WHEN THERE IS A MENU ON THE SCREEN.

M	Go directly to the MAIN menu
E	Go directly to the EDIT menu
P	Go directly to the POINT menu
L	Go directly to the LINE menu
A	Go directly to the ARC menu
O	Go directly to the OTHER CURVES menu
C	Go directly to the COPY/MOVE menu
Z	Go directly to the ZOOM menu
V	Go directly to the VERIFY DATA menu
D	Go directly to the DIMENSIONS menu
U	Go directly to the UTILITIES menu
N	Go directly to the NC,...CAM menu
S	Do an instant SAVE of the drawing
Q	Go directly to the QUIT menu
T	Go directly to the TRIM menu
R	REPAINT the screen (REDRAW)
H	Turn HELP on or off

RETURN KEY (also called the ENTER KEY): Select function from the menu.

[When the prompt line on the screen shows the character [and you press it, a different option apperars.

IMPORTANT KEYS WHEN THE CURSOR IS ON THE SCREEN

P Pick a Point on the screen
L Pick a Line on the screen
A Pick an Arc or Circle on the screen
D Pick a Dimension on the screen
SPACE BAR or left button on the mouse: Pick any Entity

C Turn Cursor coordinate display on or off (in lower left corner of screen)

DEL Key Delete the last Entity that you have drawn. (On some keyboards you have to press the '.' (the period)). NUM LOCK must be on.

[When the prompt line on your screen shows the [character and you press this character a second option appears.

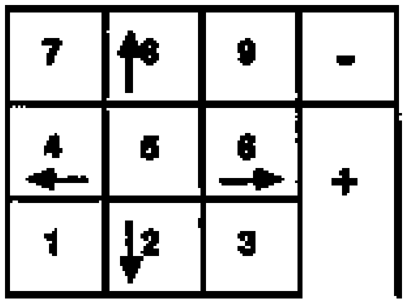
] Return to the MAIN menu

RETURN KEY or right button on your mouse:
Terminate cursor mode and return to previous menu.

BACKSPACE KEY
This key works like an "Oops key" while you are in the following functions:
SINGLE DELETE: Restore last deleted Entity
TRIM: Restore last trimmed Entities
DIMENSION: Move center of a Dimension

.....continued

CURSOR CONTROL USING THE ARROW KEYS



1,2,3,4,6,7,8,9: move the cursor around (Num Lock must be on!)

ENTERING NUMBERS

C Turn on the Calculator while entering a number. (See back of manual for a full description of the calculator)

DIGITIZING TABLET

IMPORTANT NOTE:

1. You must use the puck (mouse) with the tablet. Frequently a stylus or pen is also supplied with your digitizing tablet. Do not use the stylus, as it can be unreliable.
2. Try to keep the puck on the tablet all the time. When the puck is not on the tablet you may get a time delay between each function.

TWO FUNCTIONS:

With the tablet you can do two things:

1. With the puck you can select functions from the menus. When the cursor is on the screen you can pick Entities or indicate positions.
2. With the puck and cross-hairs on the puck you can digitize a part.

KURTA DIGITIZERS

BOBCAD will work with any size KURTA digitizer. The KURTA must be initialized when you turn on your computer or at least before you start BOBCAD. This is described in the manual that came with your KURTA. Make sure you set the KURTA to high resolution when you initialize it!

HOW TO DIGITIZE A PART FROM A DRAWING

1. Tape the drawing to the tablet. Be sure it is straight.
2. Select function CUSTOM from the UTILITIES MENU. Next select **DIGITIZE 1:1**. Now each inch on your tablet corresponds to an inch inside BOBCAD !
3. Now you can use function POINT: SKETCH or function LINE: SKETCH or function LINE CONTINUOUS: SKETCH to digitize the part as follows: Place the cross-hairs of the puck on the part and tap button 1 of the puck lightly to register each position.

CALCULATOR

Section 7

The calculator can be switched on whenever you can enter a value. For example: If you want to draw a Point at X = 1.2 + 0.3 , Y = 5 then do the following:

Go to the POINT MENU
Select function "COORDINATES"
Do not type in the value but press the C key.

The Calculator appears at the bottom of your screen and waits for your equation.

Enter the equation: 1.2 + 0.3
Next press the enter key or '=' and the answer is displayed.

If you make a mistake and do not want the value use the backspace key to make changes to the equation.

If the answer is accepable, hit the enter key again and the value is inserted as the X value. Next enter the Y value as usual.

Examples of equations the calculator will accept:

2 + 3/4 = 2.75

2 * 7 - 1.5 = 12.5

2 * (7 - 1.5) = 11

((1/2 * SIN(30) + RCOS(PI/3))/(6-2**3) = -.375

......continued

The calculator accepts all the functions shown below. The order in which the calculation takes place is the standard order. The functions are shown below in increasing order of priority. (The functions on the top of the list are calculated last).

FUNCTION		WHAT IT DOES	
( )	=	Brackets	
+	=	Add	
-	=	Subtract	
*	=	Multiply	
/	=	Divide	
**	=	Square	
SQRT	=	Square root	
PI	=	3.1415926	
ABS(xxx)	=	Absolute value	
SIN(degrees)	=	Sine of degrees	
COS(degrees)	=	Cosine of degrees	
TAN(degrees)	=	Tangent of degrees	
RSIN(xxx)	=	Sine of radian	
RCOS(xxx)	=	Cosine of radian	
RTAN(xxx)	=	Tangent of radians	
ASINE(xxx)	=	Arcsine	(answer in degrees)
ACOS(xxx)	=	Arccosine	(answer in degrees)
ATAN(xxx)	=	Arctangent	(answer in degrees)
RASINE(xxx)	=	Arcsine	(answer in radians)
RACOS(xxx)	=	Arccosine	(answer in radians)
RATAN(xxx)	=	Arctangent	(answer in radians)

OTHER KEYS:

Back space key: Correct the equation if you make a mistake.

Enter key: End of equation. Hit enter key again to accept the value.

= : Same as Enter key

PRINTER

With BOBCAD you can print a drawing in 2 ways:
1.SCREEN DUMP: You can print straight from the screen to the printer. A SCREEN DUMP is not to scale and the resolution depends on the resolution of your screen and your printer. The screen dump will not work on each computer and will not work on laser printers. In that case you can only use the next print option.
2. Through the program PRINTDGN: This program lets you put a drawing onto your printer to scale. The resolution depends on the resolution of your printer. PRINTDGN will work on dot-matrix printers and laser printers.

1. HOW TO DO A SCREEN DUMP

Set-up:

When you switch your computer on do the following:
If you have DOS 5.0 and a VGA graphics card then type the command GRAPHICS (which is part of DOS 5.0). If you have an older version of DOS then type the command EPSON (supplied with BOBCAD). The EPSON command works on EGA cards only. You can make this automatic every time you turn the computer on by putting the GRAPHICS command or the EPSON command in the 'AUTOEXEC.BAT' file using you text editor.

(Note: If you do not run the 'GRAPHICS' command or the 'EPSON' command then only the characters will appear on the printer, no graphics.)

How to run:

While in BOBCAD when there is a drawing on the screen, hold down the 'SHIFT' key and press the 'PRNT SCRN or 'Print Screen' key on your keyboard. After a few seconds what is on the screen will be dumped onto your printer.

.....continued

2. PRINTDGN

BOBCAD has a program call 'PRINTDGN'. PRINTDGN can take a BOBCAD drawing and put it on your printer to any scale with the maximum resolution your printer allows.

SET-UP:

1. PRINTDGN requires the correct driver for each printer model. Included with the software are over 100 different printer drivers. (Driver file names end with the extension '.PRT')

First try to run PRINTDGN to see what happens (see page PRINT-3). If the print comes out correctly you can ignore the following set-up procedure.

If the print does not come out at all:

a) Look for the correct driver for your printer in the list of printer drivers supplied with the software. You will find the printer drivers on the floppy disk called PRINTER DRIVERS in subdirectory PRINTDRV. If you cannot find one that matches exactly your printer try one that looks like it.

b) Copy the driver to the file 'DPRINT.PRT' in the BOBCAD directory using the copy command. (For example: go to the BOBCAD directory and type: COPY A:\PRINTDRV\EPSONH.PRT DPRINT.PRT , if you have a laser printer type: COPY A:\PRINTDRV\HPLSR100.PRT DPRINT.PRT). Next try to run the program PRINTDGN (see page PRINT-3 on how to run).

Copy A:\ ~~PRINT~~ Setup B

If the print is not to scale:

1. In BOBCAD draw a square with both sides 6" long. Put in a vertical dimension (for the Y direction using the DIMENSION MENU. Save the drawing with the name 'SQUARE' and quit BOBCAD.
2. Type 'PRINTDGN' to start the printer program. Next enter the name of the drawing (SQUARE). Next press 'C' from the menu to configure the printer scale.
3. Enter X size = 6.0
   Y size = 6.0
4. Select 2.SCALE = 1:1

....continued

Your printer will now print the square. Measure the square and write down the dimensions. If each side is exactly 6" the drawing is to scale and you are done. If one side or both sides are not to scale than:
Repeat step 2 above.
Repeat step 3 but enter:
X size = dimension you measured
Y size = dimension you measured
Repeat step 4.
The print is now to scale!

TO RUN PRINTDGN

Type 'PRINTDGN' while in DOS.
Next type the name of the drawing you want to print (without typing the extension '.DGN'). You can include a path if you need to (For example the name **\BOBCAD\ABCDXX**).
Next select the scale mode, for example: **2.SCALE 1:1**

If the print is not to scale or nothing appears on the paper, follow the set-up procedures on page **PRINT-2**. The software has been shipped to you with a standard **EPSON 9-pin** configuration. This should work on most dot-matrix printers but will not work on a laser printer. In this case follow the procedures on page **PRINT-2**.

HELP

If you need help call the technical support office at (408) 988-2345.

Copy setup-B

TECH Support
Kyle

1-408 436-7777

type

Output

INVOLUTE GEAR

The Gear function is in the OTHER CURVES MENU.

BOBCAD generates 1 gear tooth from the values the user enters. The rest of the gear must be generated from this one tooth using the COPY functions.

# TEETH	:	Total number of teeth making up the gear.
PITCH DIAM	:	Pitch Diameter.
PRESS. ANG	:	Pressure Angle at Pitch Diameter.
OUT DIAM	:	Outside Diameter of Gear.
RES. ANGLE	:	Resolution Angle: length of Arc in degrees For example set to 5. (see below for explanation)

The sliding surface of each tooth is made up of small Arcs which is the most optimal solution. The length of each Arc is set with the RES. ANGLE (Resolution Angle).

The tooth BOBCAD generates consists of the following:
1. The Gear is centered around X = 0, Y = 0
2. Several Arcs make up the sliding surface.
3. There are 4 Arcs with center X = 0, Y = 0. The first has the outside diameter, the second has the pitch diameter and is dashed, the third and fourth have the standard inside diameter.

BOBCAD generates only 1 tooth so the user can complete the tooth any way he needs before using the COPY:ROTATE function to draw the whole Gear. For example you can:

1. Draw a Fillet between the "Arc with the outside diameter" and each of the top corners.
2. Or replace the ouside Arc with a Line and then you may want to draw a Fillet at each of the top corners.

...continued

3. Use the function 3 ENTITIES in the ARC MENU to draw an Arc connecting one tooth to the next and the inside radius. (Note you must first draw another tooth using the COPY/ROTATE function). Or you can decide to draw a Line from the first Arc to a Point X = 0, Y = 0. Next draw a Fillet between the Line and the "Arc with the inside diameter".

NOTE: The first and smallest Arc of the sliding surface may be too small for your controller to be useable. In that case delete it on both sides of the tooth and extend the next Arc.

BOBCAD GOLD

Section 8

LATHE FUNCTIONS

BOBCAD GOLD Features

Machineable Fonts

This version offers several machineable fonts.

To draw text, select function TEXT/FONTS in the Dimensions menu. CAD TEXT is a font that is meant for putting the notes on the drawing that are not going to be machined. CAD TEXT is stored as a single element and does not use much memory. CAD TEXT is available in ISO Font only.

To change the size and angle of CAD TEXT, select the function ATTRIBUTES in the Dimensions menu.

CAD TEXT can be changed to machineable text by selecting the PATH TEXT function in the Other Curves menu. Then in the Milling menu, start machining in the lower left corner of the text and then use the function AUTO. In the Edit menu, the function SIMULATE can be used to see the actual finished part.

Using Fonts:

Select the function TEXT/FONTS from the Dimensions menu.
Next select the function FONTS.
Type in the text.
Result - the machineable font is drawn on the screen. Each character consists of many short lines. Characters are one inch high and the text starts at X = 0, Y=0.
To change the size and position of the text, use the functions in the Copy/Move menu.
To machine the fonts use the function AUTO in the CAM menus. Position and size options wil be added in the future.

Ruled Surface

A ruled surface can be drawn between more than two sets of curves using the function RULED in the Surface menu (3-D menu). The function AUTO RULED is limited to two curves only. To become familiar with the RULED SURFACE function, do the RULED SURFACE exercise in the CAM Exercises section of your BOBCAD Manual.

To draw a ruled surface between many curves, each curve must first be interpolated. An interpolated curve consists of many short lines. Each curve must have the same number of lines.

Simulate

Function simulate shows what material will be cut away based on tool size. You can select function simulate from the draw path menu (see utilities menu) and also in the edit menu (see Cam menu).

Cutall

The CUTALL feature is located in the Cam menu. When you select CUTALL lines and Arcs will be cut. Function CUTALL starts off by moving to the nearest element and then chaining around till it can go no further. Then it rapids over to the to the next nearest element and does the same thing. When everything has been cut, the tool goes back to X=0 , Y=0.

Cutall does not check for direction therefore this function is not usefull for every kind of job. To compensate for your cutter diameter you can use the OFFSET function in the Other Curves menu. Then before you use functions Cutall, make sure to BLANK the original shape.

LATHE

BOBCAD-GOLD is a CAD-CAM system that can program both Milling machines and Lathes. When you start BOBCAD-GOLD it starts up in Milling mode. To switch to Lathe mode go into the UTILITIES MENU and select function LATHE = OFF so it switches to LATHE = ON. First however you should become familiar with the software by doing the CAD exercises and the CAM exercises with LATHE = OFF. Then when you know how to work the software switch to LATHE = ON.

When you set LATHE = ON you will now be asked if you want to load the LATHE TOOL LIBRARY. Answer YES. The LATHE TOOL LIBRARY consists of a selection of pre-drawn tools. Later you can draw your own tools and add these also to the tool library.

MILLCFG

Additionally the NC output format must be set using the MILLCFG routine. In MILLCFG set STANDARD X,Y OUTPUT to LATHE OUTPUT. (LATHE OUTPUT has various options including radius and diameter programming)

When you are in LATHE mode you will find the following:

1. In the CAD the vertical axis is now the X-Axis and the horizontal axis the Z-Axis. Input for the vertical axis is in diameter (for example look at function POINT: COORDINATE or LINE COORDINATE).

2. In the OTHER CURVES MENU the POCKET function is replaced with a LATHE ROUGHING function

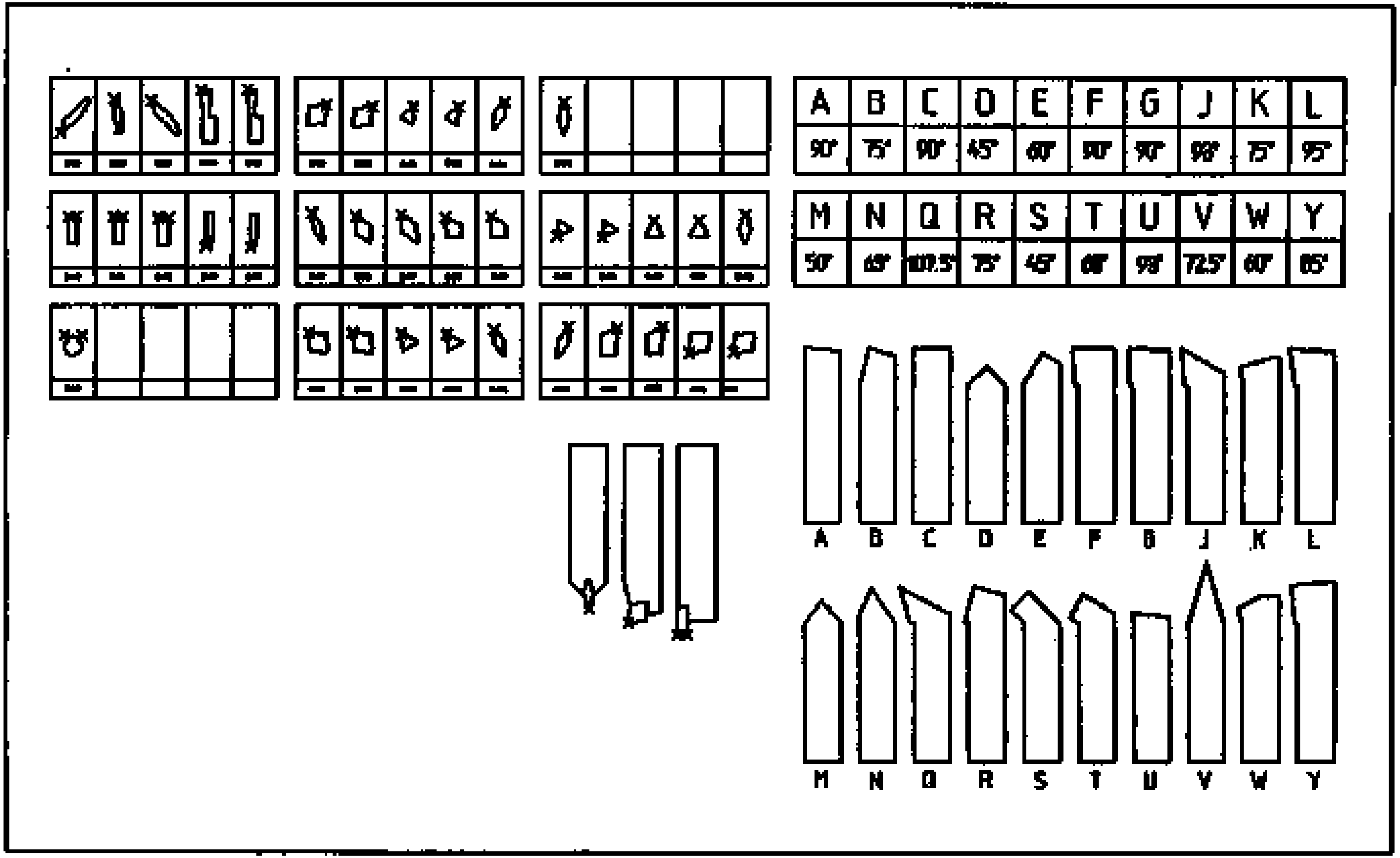
3. The CAM section of BOBCAD allows you to program X, Z, I, K output with optional R for Radius. To set this you have to run the MILLCFG program and select LATHE OUTPUT instead of STANDARD X Y OUTPUT. We have preset the output for you in file CAM-2.CFG which you can select when you go into the CAM section of BOBCAD.

4. A TOOL LIBRARY which is shown on the next page can be used in which the different tools are stored. The selected tool will be used to calculate the roughing path (pre-turn). The selected tool is also displayed in the CAM section of the software.

LATHE TOOL LIBRARY

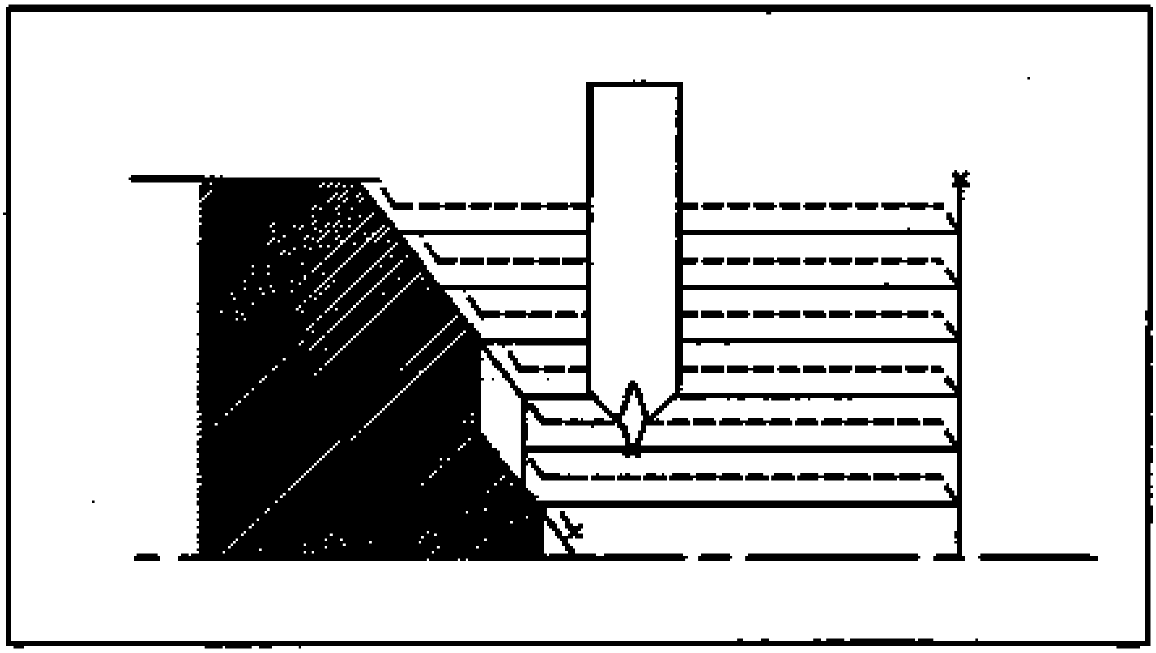
A drawing called 'TURNTOOL.DGN' is supplied with BOBCAD which has many standard tools and holders already predrawn. You can change this drawing by deleting what you don't need, adding shapes of your own tools and then saving the drawing the same way you save any drawing.

Every time you set LATHE = ON in the UTILITIES MENU you will get the question "DO YOU WANT TO LOAD THE TOOL LIBRARY ?" If you answer YES then TURNTOOL.DGN will be loaded automatically.



What to use the lathe functions for.

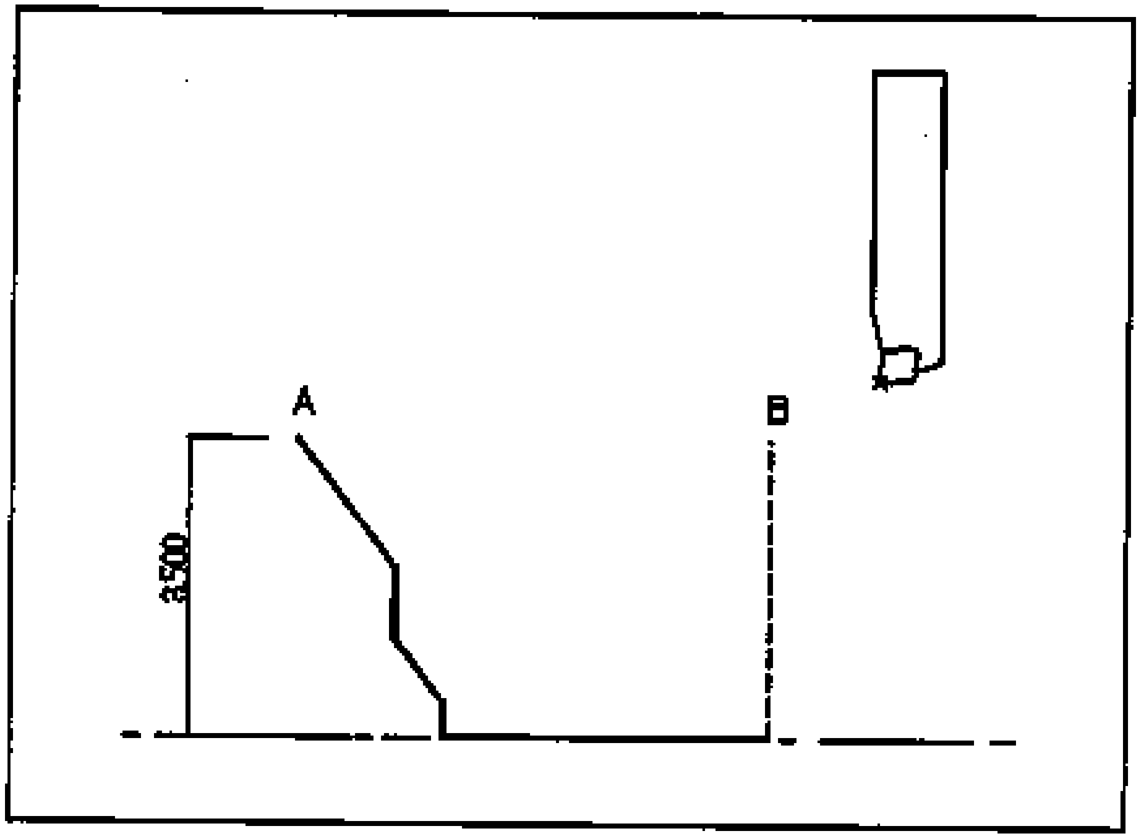
This lathe program makes it easy to program outlines of shapes for turning. Most new lathes have a lot of canned cycles already built-in. You can put the commands for these canned cycles in the CUSTOM MENUS (see EDIT MENU in the CAM MENUS). For example you can do this with a THREADING command. BOBCAD will not graphically display these canned cycles. Most lathes already have ROUGHING cycles built-in. BOBCAD also provides a roughing function. The ROUGHING function calculates a path which checks for tool collision with the part. This is shown below. The following exercise shows you how this is done. In addition the OFFSET function in the OTHER CURVES MENU can be used to generate an Offset curve which compensates for the tool tip radius.



USING THE ROUGHING FUNCTION

First in the UTILITIES MENU set LATHE = ON

Draw the outline as shown below. Make sure point A and point B are at the same height. The angle at which the roughing takes place is the same as the angle of a line between point A and point B. The Line on the right is drawn dashed. No tool interference is calculated for a dashed Line.



Next select function ROUGHING in the OTHER CURVE MENU.

Next pick the outline starting with the Line at point B (if you start at point A the cutting will go from left to right).

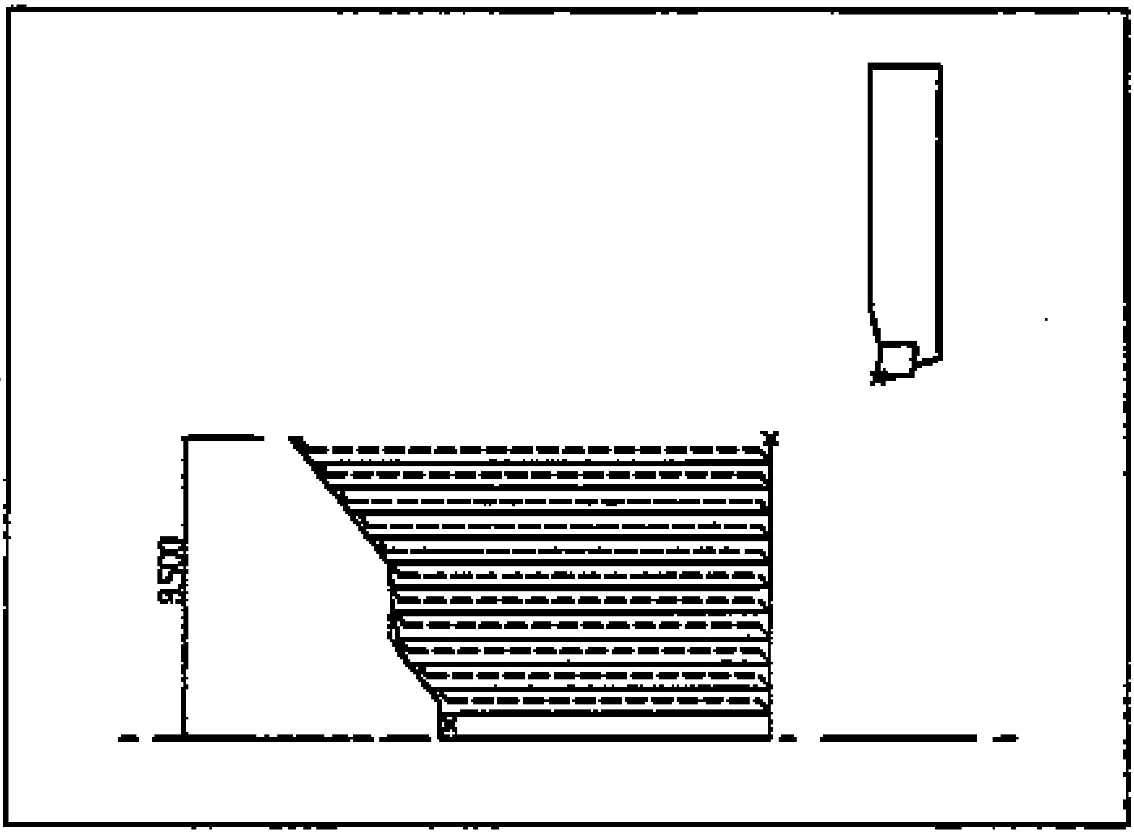
Next pick the shape of the tool. Note: the tool shape must be a closed contour.

Next pick a reference point on the tool. This Point is the extreme Point of contact between the part and the tool during setup.

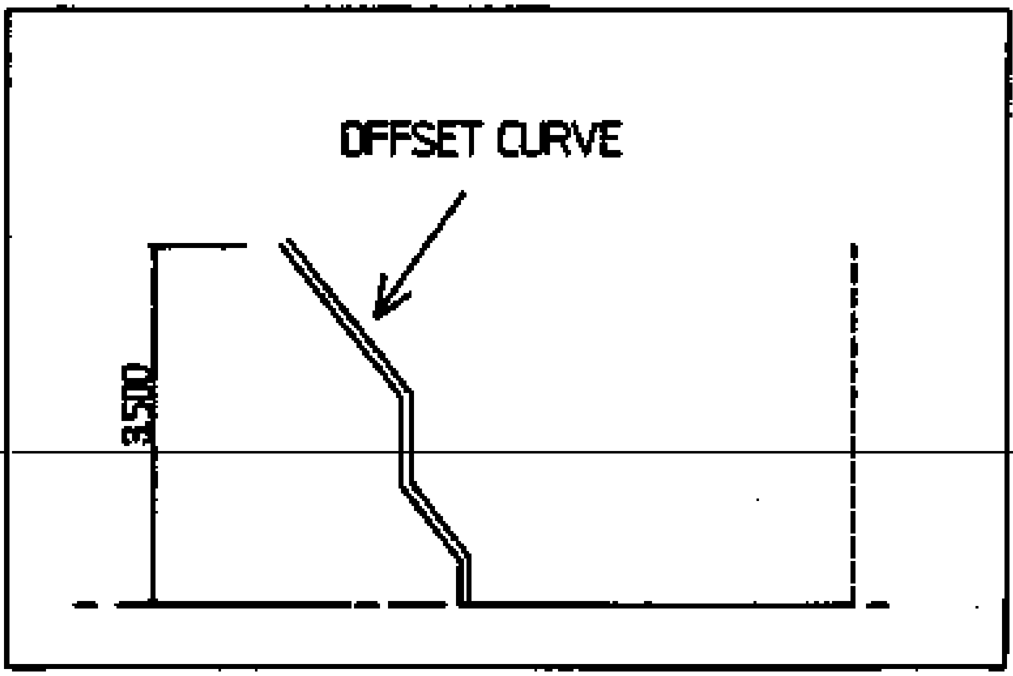
Next enter the step-over distance.

Automatically the pre-turning path is calculated which is shown on the next page. (The calculations can take a fair amount of time, the computer is not hanging up).

When you are in the CAM menus the upper right Point in the drawing is where you start machining.



To compensate for the tool tip radius without using G41 or G42 you can use the OFFSET function in the OTHER CURVES menu to create the tool center path. This is shown below. Next in the CAM menus use this Offset curve to create the NC code.



Lathe Exercise

CAD SECTION

Start a new drawing in BOBCAD and give it a name. such as LATHE1.

From the MAIN MENU, choose ZOOM, WINDOW.

Pick the first corner of the window at X = 1, Y = -1. Move the box to the upper left and finish the window at X = -4, Y = 3.

Return to the MAIN MENU, UTILITIES. Move down to LATHE-OFF and toggle this to LATHE-ON. Do not load the Tool Library at this time.

This Part will be drawn in a position that lends itself well to lathe machining. Parts should always be drawn in this manner in the Lathe portion of BOBCAD, but the COPY/MOVE feature can always be used to change the position of the part.

Remember, the lathe coordinate system is now on. The vertical axis is now the X-Axis and the horizontal axis is now the Z-Axis. Values entered as X coordinates will be seen as diameter values.

Draw the Part:
From MAIN MENU, choose OTHER CURVES, RECTANGLE, COORDINATE.

Enter X = 0, Z = 0. Choose ABS X & Y.

Enter X = .750, Z = -1.125 for the opposite corner. Corner radius = 0.

There should now be a rectangle on the screen at maximum part coordinates.

Return to the MAIN MENU,choose ZOOM. VIEW ALL. The part should occupy most of the screen.

Review the following:
1. The bottom line of the rectangle is really part center.
2. The top line is at X = .375 (A .750 Diameter value has a radius distance of .375)
3. The VERIFY DATA selection from the MAIN MENU will still measure the part in the X & Y coordinate system. Keep this in mind when using VERIFY DATA.

From ,MAIN MENU choose **LINE, COORDINATE.**

Line C	Line D
Enter XS = .5	XS = .75
ZS = 0	ZS = -.875
XE = .5	XE = 0
ZE = -1.125	ZE = -.875

The drawing should look like Figure 1

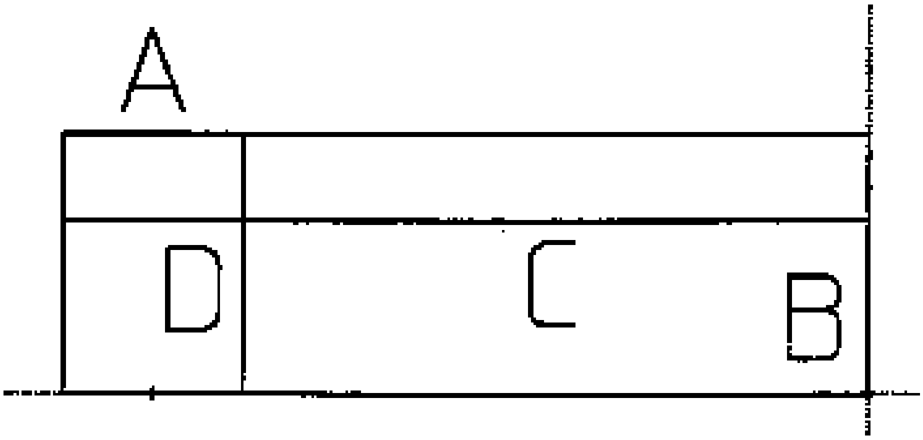


Figure 1

From MAIN MENU, choose **EDIT, TRIM, TWO ENTITY.**
Pick the upper part of line D and the right-hand portion of line C.

Select **ONE ENTITY** from the **TRIM MENU.**
Pick the lower portion of line B and then pick the point where line C intersects line B. Now pick the left-Hand portion of line A and then pick the point where line A intersects line D.

The drawing should look like Figure 2.

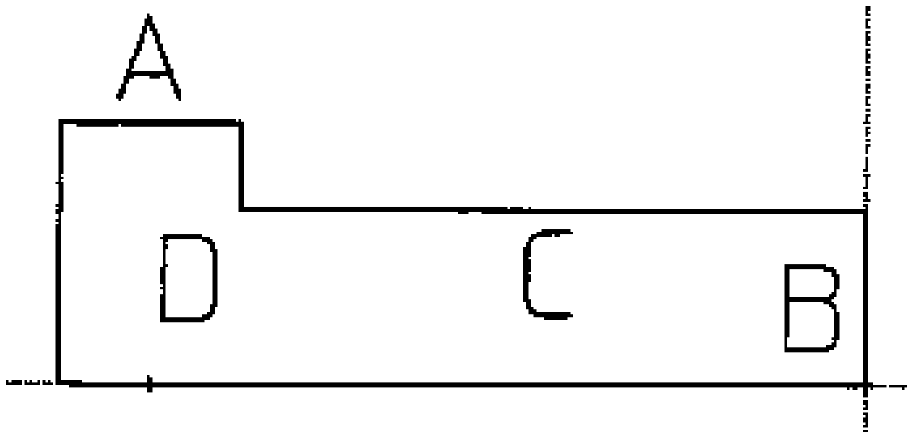


Figure 2

From **MAIN MENU**, choose **LINE,.CHAMFER, DIST/ANGLE.**
Enter a distance of .02 and an angle of 45. Now pick line C then pick line B. A chamfer should appear at corner BC. Put a chamfer at the corner of AD in the same manner.

From **MAIN MENU**, choose **ARC, FILLET.**
Pick line D, then pick line C. then enter a radius of .075

The drawing should look like Figure 4.

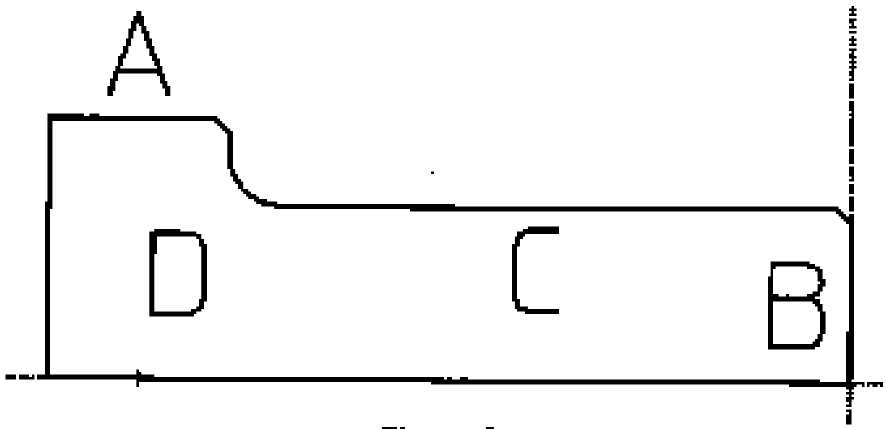


Figure 3

Figure 4

Press **S** to SAVE the drawing.

Create the Finish Path:

To create the finish pass, the **OFFSET** command in the **OTHER CURVES MENU** will be used to build a path parallel the finished part. This parallel path will be centered on the center of the the tool nose radius to keep the edge of the tool tangent to the part at all times.

The Tool Length Offsets for the finish pass on the LATHE need to be adjusted by the radius of the tool nose in the Z axis. Depending on the LATHE, the tool length offset should be adjusted by the Tool nose radius in the X-axis for a radius machine, or by twice the tool nose radius for a diameter machine.

To keep things simple, different layers will be used for the machining of the finish pass and the rough pass.

From the **MAIN MENU**, choose **LAYER, NEW**
Type in the new layer number as **1.**
From the **MAIN MENU**, choose **OTHER CURVES, OFFSET**, CHAIN, and **DISTANCE**
Enter the Distance as .0315

Select line B (closest to the X0 end of the line), indicate the direction of the arrow UP and place it to the right side of the part edge, then select the line A as the last entity. Press the right mouse button. The Finish path is now drawn.

Now let's add a start point and a finish point to the finish path. This makes it easier to control how the tool approches and exits the part.

From the **MAIN MENU**, choose **POINT MENU, END**
Pick the bottom end of the finish path line parallel to line B and the left end of the finish path line parallel to line A
Press the right mouse button, then pick **INC X-Y**, pick the first point just put in , **INC X RAD** = 0, Z = .1. Pick the second point, **INC X RAD** = .1, Z = 0.

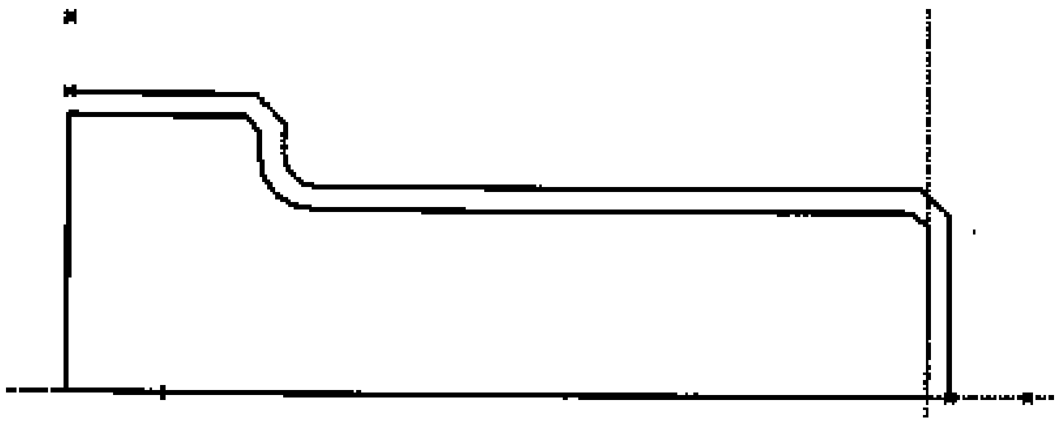


Figure 5

Create the Rough Path
The rough pass will be generated from the finish pass on layer 3.
Before that we will build the roughing boundry on layer 2 and leave .030 stock on the part for the finish pass to remove. Yes, that is a lot of material to leave for the finish pass, but when you see how to build the roughing boundry, you can make that stock amount smaller.

Since the ROUGH routine works with the theoritical tool tip, the Tool Length Offsets would NOT be adjusted like the finish pass.

Let's load the TOOL LIBRARY now.
From **MAIN MENU**, choose **UTILITIES, LATHE-ON, LATHE OFF**
Load tool Library? **YES**

Use **ZOOM, VIEW ALL** to see all of the available tools. Many common shapes are represented and there is a choice of cutting edge radius for most of the tools. Use the **ZOOM WINDOW** to get a closer look at the tools. Notice each tool has a point on the cutting edge for reference. The **COPY/MOVE** option from the **MAIN MENU** can be used to move any tool in the library to any point on the screen, but this option will not be used in this exercise.

From **MAIN MENU**, choose **LAYER, NEW.**
Enter the new layer number as 2.
Press the Right Mouse Button.
Choose **BLANK, PICK&MATCH.**
Select the finish pass. This blanks the finish pass from the screen so that it cannot be changed accidentally.

Let's build up the roughing boundry.
From the MAIN MENU, choose OTHER CURVES, OFFSET, CHAIN, DISTANCE.
Enter the Distance as .03.
Select line B (closest to the X0 end of the line), indicate the direction arrow UP and to the right side of the part edge, then select the line A as the last entity. Press the right mouse button to indicate that we are done selecting the shape.
Pick YES for the question SELECTION OK? and NO for CONTINUE OFFSETTING NEW SHAPE?

From the MAIN MENU, choose LINE, PARALLEL.
Pick the Finish pass line parallel to line B, indicate the right side of the line, Distance = .1 and pick the finish pass line parallel to line A, indicate the side as above the line, Distance = .1.
Return to the MAIN MENU and choose ZOOM, REDUCE.
Pick LINE, JOIN and join each of the parallel lines with the original lines as shown in Figure 6.
Hit the Right Mouse Button.
From the MAIN MENU, choose EDIT, TRIM, ONE ENTITY.
Select the far right hand line, near the top end of the line, then select the upper most horizontal line.
Hit the Right Mouse Button

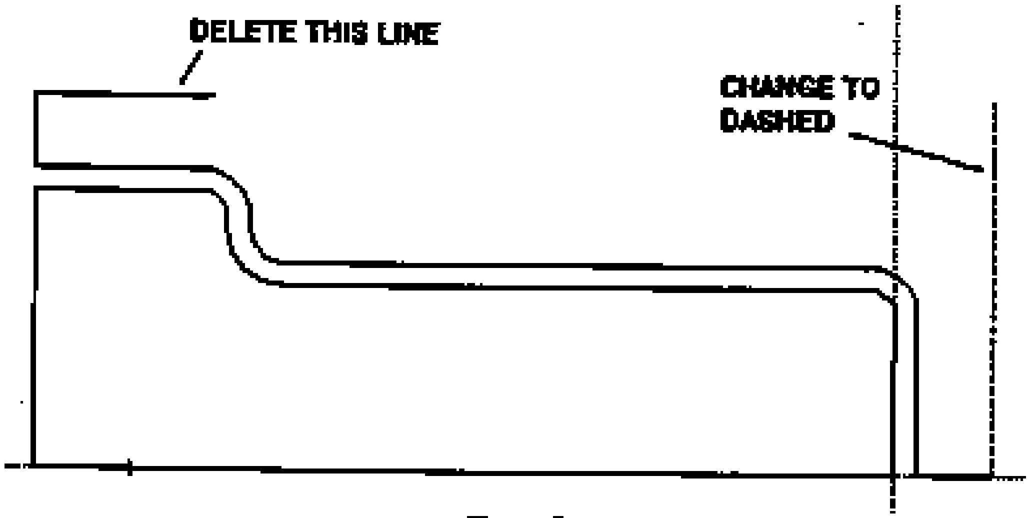


Figure 6

The LATHE ROUGH routine will create the rough passes parallel to the angle created by the the upper most end of the rough boundry, so for this part we want the lines to be at the same X diameter so that the rough passes will be parallel to the X axis.

From the MAIN MENU, choose EDIT, DELETE, SINGLE
Delete the line indicated in Figure 6. Press the Right Mouse Button when done. If you picked the wrong line press the BACKSPACE key. This will undelete the last deleted entity.

One last thing to do before we rough the shape is to change the far right hand line from a solid to a dashed line. The LATHE ROUGH routine looks at the whole tool to make sure it does not bump into the part. If we leave this line as a solid line then the tool will stay away from the right side of the roughing boundry. We don't want that!. The left side is all right, but we want the referance point of the tool to go all the way to the far right hand line to clean all the material off the part before we come in with the finish pass.

From the **MAIN MENU**, choose **COLOR/STYLE, MODIFY STYLE, SINGLE**
Pick the far right hand line, press the Right Mouse Button, then choose **DASHED**

Press S to **SAVE** the drawing

From the **MAIN MENU**, choose **LAYER, NEW**
Type in the new layer number as 3.

From the **MAIN MENU**, choose **OTHER CURVES, LATHE ROUGH**

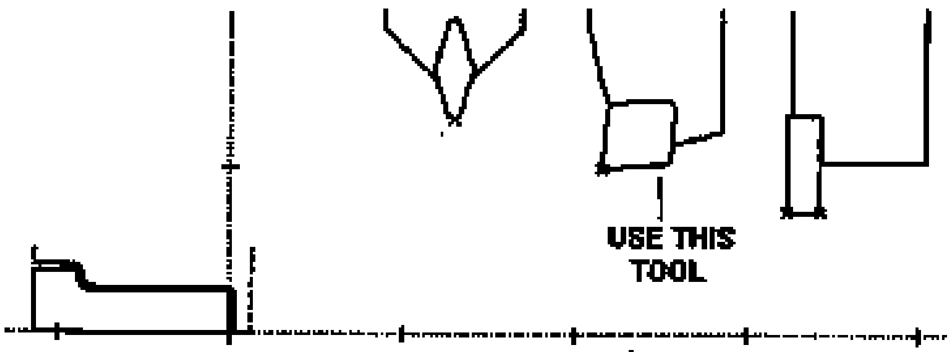


Figure 7

Select the right vertical line, point the arrow down, press the] key to complete the chain, and choose YES for SELECTION OK?

Select the tool shown in Figure 7 by pressing Z for the **ZOOM MENU, VIEW ALL** then press the Right Mouse Button (Note: don't press the ESC key, this will knock you out of the current command). Pick the tool holder and the point at the tip for the tool referance point.
Enter the Distance between cuts as .075
The lathe rough pass will be built.

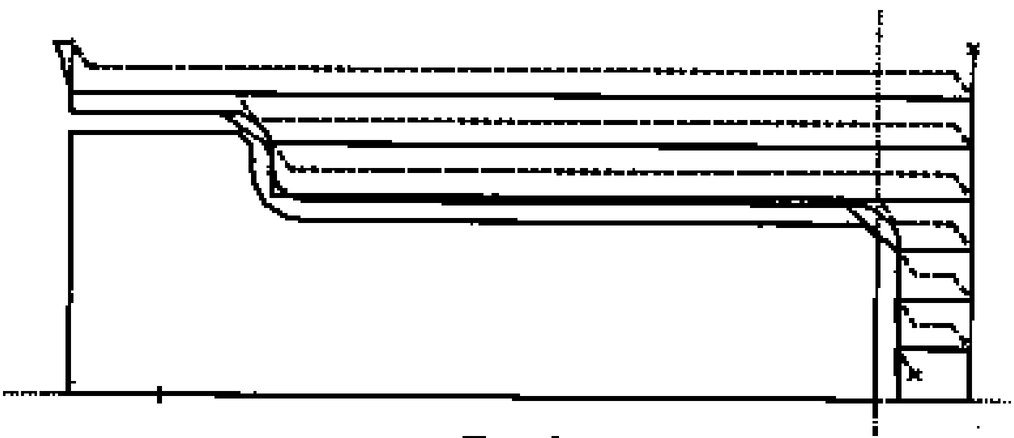


Figure 8

The CAD section of this exercise is now complete. The CAD portion of BOBCAD may be entered again at any time, if changes to the drawing need to be made.

It is important to blank out the roughing boundry before we go to the CAM section to produce the machine G-code. Otherwise the boundry will get in the way of selecting the first cutting line of the roughing pass.

From the **MAIN MENU**, choose, **LAYER**, **BLANK**, **LAYER**
Type in **2** to blank layer 2 (our roughing boundry layer)

CAM Section

Cutting the Part

Select **NC CAM** from the **MAIN MENU**, then select **NEW FILE**. Give the file a name, such as **LATHE1**. Pick **CAM-1**.CFG and notice the first few lines put in during configuration will appear along the bottom of the screen.

Select **PICK TOOL** from the **CAM MENU**. Pick the same tool used for roughing.
Note the tool's change in color when it is chosen.
Be sure #8 in the CAM MENU is in **RAPID MODE**.

Select **POINT MOVE** and pick the point in the upper right of the roughing pass. The tool will make a rapid move to the new location (indicated by the dashed line).

Toggle #8 in the CAM MENU to **FEED MODE**.

Select **AUTO** and pick the vertical line beneath the point. Move the mouse so that the arrow points down in the direction of the rough pass and press the left button on the mouse. Hit the] key. The tool should move through the entire roughing pass. Note the code for each move is shown at the bottom of the screen.

Select **EDIT**, **TOOL CHANGE**
Add a tool change block for the next tool.

Press **M** for the **MAIN MENU**, Choose **LAYER**, **BLANK**, **LAYER**
Type in **3** to blank the rough pass, then choose **UNBLANK** and type **1** for the finish pass.

Go back to the NC...CAM, you will continue right where you left off.

Make sure #8 in the CAM MENU is set to **RAPID MODE**.

Select **POINT MOVE** and pick the far right hand pointon the finish pass.

Toggle #8 in the CAM MENU to **FEED MODE**.

Select **SINGLE** and pick the line above the point. Move the mouse so that the arrow points in the direction of the uncut part and press the left button on the mouse. The tool should move up the line. Note the code for this move is shown at the bottom of the screen.

Continue using **SINGLE** to move along the part.
When the tool gets to the end of the last line, change **FEED MODE** to **RAPID MODE**. Go to **POINT MOVE** and select any point off the shape to get the tool away from the part.

Press S to save the drawing.

Creating Your Own Tool

If the user chooses not to use the tooling library, any tool shape can be drawn in the CAD section of BOBCAD. An example is shown in the following steps.

Select **POINT** from the **MAIN MENU**, then select **COORDINATE**. Enter X = 1.5, Z = 1.

Using **WINDOW** in the **ZOOM MENU**, put a one inch window around the point, with the point located in the lower left corner of the window.

Using the **LINE** function, draw a line from **X = 1.5, Z = 1** to **X = 2.5, Z = 1**

Using **PARALLEL** in the **LINE MENU**, make a parallel line **.092** inches to the right of the line just drawn. Stay in the **LINE MENU** and use **JOIN** to connect the top and bottom of these lines.

You now have a basic example of a tool that can be used in the CAM section.

FreeDOS Spec

Appendix, Commands Behavior

DOS commands may be internal or external. Any DOS command may be implemented internally by the shell, or provided externally as in a program. If external, the program may be either a COM or EXE, with the exception of COMMAND.COM.

In each of the following, d: represents a drive letter, path represents a full pathname, and file represents a full filename, or a file mask including * and ?.

For all commands, internal and external, the command line option /? should generate a help screen, and the program should immediately exit. The exit code is not defined, but we strongly suggest an exit code of zero.

A few traditional MS-DOS commands have been dropped from the Spec. Dropping from the Spec does not mean that such a program will be ignored if one ever comes along, but this does mean that we will not actively seek out to develop a new version. This is because they were considered as part of a legacy migration thing for MS-DOS, which the FreeDOS Project is not interested in carrying forward. Where a program has been dropped from the Spec, an annotation has been made to explain why.

---

append

Allows programs to open data files in the specified directories as though they were local to the working directory.

[Dropped, because the program was considered to be a “crutch” for legacy DOS-based applications as DOS moved to support directories. Append was used primarily to support programs that did not know about directories on a DOS system.]

```
append d:]path[;…? [options]
```

Options:

```
;
When used by itself, the semicolon cancels the previously specified
appended directories.

/x[:on|:off]
Allows the shell (if supported) to search these directories as though
```

```
they were in the PATH.  Default is /x:off.  /x is the same as /x:on.

/path:on
Specifies if DOS is to search appended directories for a program's
data file, even if the program specifies a full path.  Default is
/path:on.

/e
Add the list of appended directories to the PATH.

If no paths are named, display the current appended directories.
```

assign

Reassigns disk operations from one drive to another. For example, to reassign disk operations for drive A: to drive B:.

```
assign [d:=d:] […]
assign /status
```

Options:

```
/status
Displays the current assignments.

If no drives are named, display the current assignments.
```

attrib

Change a file's attributes.

```
attrib [options] [[d:][path]file]
```

Options:

```
+r | -r
Sets the read-only attribute.  -r will clear it.

+a | -a
Sets the archive attribute.  -a will clear it.

+s | -s
Sets the system attribute.  -s will clear it.
```

```
+h | -h
Sets the hidden attribute.  -h will clear it.

/s
Act on subdirectories, too.

If no file is named, display the attributes for all files in the
current directory.
```

break

Sets or clears the Break or Ctl-C check.

```
break [on|off]
```

Options:

```
on | off
Turn on (or off) the Break check.

If no arguments are named, display the current Break check status.
```

cd, chdir

Displays the name of the current directory, or changes to another one.

```
cd [d:][path]
chdir [d:][path]
```

chcp

Change the DOS code page.

```
chcp [nnn]
```

Options:

```
nnn
The code page that is to be loaded.  Use 437=US, 850=Multi (Latin
I), 852=Slavic (Latin II), 860=Portuguese, 863=Canadian-French,
865=Nordic.

If no code page is named, display the current code page.
```

chkdsk

Check the disk for errors.

```
chkdsk [d:] [options]
```

Options:

```
/f
Fix errors on the disk when found.

/v
Display the name of every file as the disk is checked.

If no drive is specified, check the current disk for errors.
```

cls

Clear the screen.

```
cls
```

choice

Suspend processing, and wait for the user to press a valid key.

```
choice [options] [text]
```

Options:

```
/C[:]choices
Specifies allowable keys. Default is YN

/N
Do not display choices and ? at end of prompt.

/S
Treat choices as case sensitive.

/T[:]c,nn
Specify the default choice `c' after `nn' seconds of no response.

text
Prompt string to display (Default=none.)
```

command

Start a new command shell.

```
command [[d:]path] [device] [options]
```

Options:

```
d:path
The drive and path where the shell is to look for the transient part
of the program.  This is usually only needed to set the COMSPEC.

device
An alternate device for all i/o.

/e:nnnnn
The environment size, in bytes, in the range 160-32768.  This number
will be rounded up the nearest 16 bytes.  The default is 256.

/p
Make the shell permanent, so the EXIT command does not exit the
shell.

/c {string}
Execute the command in {string}, then exit.

/msg
Load any error messages that might be stored on disk into memory.
```

comp

Compare two files.

```
comp [d:][path]file [d:][path]file [options]
```

Options:

```
/d
Display differences in decimal format.  Default is hex.

/a
Display differences using characters.
```

```
/L
Display the line number on which the difference occurred.

/n=nnn
Compares the first nnn lines of both files, even if the files are of
difference size.

/c
Ignore case.
```

copy

Copy one or more files.

```
copy [/a|/b] [d:][path]file [[/a|/b] +[d:][path]file] […] [dest] [/v]
```

Options:

```
d:path\file
The name of a file to copy.  This is the source.

dest
The name of the files to copy to.  This is the destination.  If the
dest filename contains a *, then replace this part of
the filename with the same part as the source.

/a
Copy an ASCII file.  If this precedes a filename in a list of files,
then from this file on, copy as ASCII files.

/b
Copy a binary file.  This is the default. If this precedes a
filename in a list of files, then from this file on, copy as binary
files.

/v
Verify that the files are written correctly.
```

ctty

Change the terminal device for your DOS session.

```
ctty device
```

Options:

```
device
The device to use.  Valid devices are prn, lpt1, lpt2, lpt3, con,
aux, com1, com2, com3, com4.
```

date

Display or change the date.

```
date [date]
```

Options:

```
date
The new date for your system.  Values for the day, month, and year
must be specified, and values may be specified by periods, hyphens,
or slashes.  Either a 4-digit or 2-digit year may be used.  If a
2-digit year is specified, the year must be in the range 00-99.  If
the year is xx:00-79, assume 20xx.  If xx:80-99, assume 19xx.

If no date is given, display the current date and prompt for a new
date.
```

del, erase

Delete files.

```
del [d:][path]file [options]
erase [d:][path]file [options]
```

Options:

```
d:path\file
A file to delete.  A period may be used to specify all files in the
current directory, and is the same as *.*.

/p
Prompt before deleting the file.
```

deltree

Delete an entire directory tree, including subdirectories.

```
deltree [options] [d:]path
```

Options:

```
/Y
Yes, delete without asking.

d:\path
The directory to delete, including subdirectories.
```

dir

Displays the contents of a directory.

```
dir [d:][path][file] [options]
```

Options:

```
d:path\file
A specific file to display.

/p
Display one page of the listing at a time.

/w
Display the listing in wide format.

/a[:attribs]
Display only the files and directories with the specified
attributes.  Default is /a:hsdar.  You may use the following for
attribs:
  h | -h
  Hidden files.  -h for files that are not hidden.

  s | -s
  System files.  -s for files that are not system files.

  d | -d
  Directories.  -d for files.
```

```
  a | -a
  Files with the archive bit.  -a for files without the archive bit.

  r | -r
  Read-only files.  -r for files that are not read-only.

/o[:order]
Display the listing in the specified order. You may use the
following to specify the sort order:
  a | -a
  Sort alphabetically A-Z.  -a to sort Z-A.

  e | -e
  Sort by extension A-Z.  -a to sort by extension Z-A.

  d | -d
  Sort by date and time.  -d to sort in reverse order.

  s | -s
  Sort by size, smallest-biggest.  -s to sort biggest-smallest.

  g | -g
  Group directories first.  -g to group directories last.

/s
List every occurrence in all subdirectories.

/b
Bare format.  List one name per line.

/L
Display names in lowercase.
```

diskcomp

Compares the contents of two floppy disks, track by track

```
diskcomp [d: [d:]] [options]
```

Options:

```
d:
The drive letter for a floppy disk.  If only one drive letter is
given, assume the working drive (if a floppy drive.)  If no drive
letters are given, assume both are the working drive (if a floppy).
If both drives are the same, diskcomp will read one floppy at a
time, and you may need to swap floppies.

/1
Compare only the first sides of the disks, even if double-sided
disks.

/8
Compare only the first 8 sectors per track.
```

diskcopy

Copy the contents of a floppy disk to a second floppy.

```
diskcopy [d: [d:]] [options]
```

Options:

```
d:
The drive letter for a floppy disk.  If only one drive letter is
given, assume the working drive (if a floppy drive.)  If no drive
letters are given, assume both are the working drive (if a floppy).
If both drives are the same, diskcopy will read one floppy at a
time, and you may need to swap floppies.

/1
Copy the first side only, even if double-side.

/V
Verify that the data is copied correctly.
```

echo

Displays a message.

```
echo [message]
```

edit

Starts the DOS editor.

```
edit [[d:][path]file] [options]
```

Options:

```
/B
Use a black-and-white (mono) display.

/G
Use faster update for a CGA screen.

/H
Display using the highest video/text resolution available.

/NOHI
Do not use high-brightness colors.
```

emm386

The expanded memory manager (EMM) for ‘386 systems (or better)

```
emm386 [on|off|auto] [w=on|w=off]
```

Options:

```
on | off
Turn expanded memory support on or off.  Default=on.

auto
Only support expanded memory when a program asks for it.

w=on | w=off
Enables or disables support for the Wietek coprocessor.
Default=off.
```

exe2bin

Converts EXE programs to binary format. This is a software developer’s tool.

```
exe2bin [d:][path]file.exe [d:][path]file.bin
```

fastopen

Decreases the amount of time needed to load files and programs.

[Dropped, because disk cache should be (eventually) implemented in the kernel, not by an external program. Also, Fastopen was originally created to increase DOS application performance on systems with slow drives, where today disk drives are much faster.]

```
fastopen d:[]nn] [d:[[nn]] …. [options]
```

Options:

```
d:
The drive on which to use fast-open.

nn
The number of files that can be tracked, in the range 10-999.
Default=48.

/X
Create the cache in expanded memory instead of conventional memory.
```

fc

Compare two files.

```
fc [options] [d:][path]file1 [d:][path]file2
```

Options:

```
/A
Abbreviate the ASCII output.  Instead of displaying all lines, fc
will display the first and last lines that are different.

/C
Ignore the case of letters.

/L
Compare in ASCII mode.  Displays all lines that differ.  This is the
default mode for files that are not EXE, COM, SYS, OBJ, LIB, or BIN.

/LBn
Sets the number of lines for the internal buffer.  Default=100.  If
```

```
the files have more than this many differences, fc will quit.

/N
Display line numbers.

/T
Do not expand tabs to spaces.  Default is to expand tabs to 8-space
stops during comparison.

/W
Compress white space during comparison.

/nnn
Specifies the number of lines that must match after a miscompare for
the files to be resynchronized.  Default=2.

/B
Compare in binary mode.  Does not attempt to resynchronize after a
miscompare.
```

fdisk

Configures a hard disk.

```
fdisk
```

find

Displays lines in a text file that contain a string.

```
find [options] string [d:][path]file
```

Options:

```
/V
Invert the search.  Display lines that do NOT contain the string.

/C
Only display a count of the matching lines.

/N
Display line numbers.
```

```
/I
Ignore case during the comparison.
```

format

Format a hard drive or floppy disk.

```
format d: [options]
```

Options:

```
d:
The hard drive or floppy disk to format.

/V:label
Assign this label to the formatted disk.

/Q
Do a quick format.  This clears the FAT and root directory, but does
not erase the disk.

/U
Do an unconditional format.

/F:size
Format the disk to a specific size.  Valid values are:
  160 | 160k | 160kb
  Format to 160kb.

  180 | 180k | 180kb
  Format to 180kb.

  320 | 320k | 320kb
  Format to 320kb.

  360 | 360k | 360kb
  Format to 360kb.

  720 | 720k | 720kb
```

```
  Format to 720kb.

  1200 | 1200k | 1200kb | 1.2 | 1.2m | 1.2mb
  Format to 1.2MB.

  1440 | 1440k | 1440kb | 1.44 | 1.44m | 1.44mb
  Format to 1.44MB.

  2880 | 2880k | 2880kb | 2.88 | 2.88m | 2.88mb
  Format to 2.88MB.

/B
Reserve space to make a bootable disk.

/S
Make a system disk (copy files to make it bootable.)

/T:tracks
Specify the number of tracks on the disk.

/N:sectors
Specify the number of sectors.

/1
Format a single-sided disk.

/4
Format a 5-1/4 inch 360KB DSDD disk on a 1.2MB drive.

/8
Format a 5-1/4 inch disk with 8 SPT.
```

graftabl

Load the graphics table for a specific code page.

[Dropped, because most monitors can display extended characters (128–255) without Graftabl. This command was originally introduced by MS-DOS to support the extended character set on systems that were not originally equipped to display them properly. While some users may miss this command, we feel there are not enough systems that still need Graftabl to justify its inclusion in the Spec.]

```
graftabl [nnn] [options]
```

Options:

```
nnn
The code page you want to load.

/STATUS
Display the code page that was loaded, or is being loaded.
```

graphics

Allow Prtscr to print graphics screens.

```
graphics [[d:][path]file] [type] [options]
```

Options:

```
d:path\file
The full pathname of the printer profile.

type
The printer type.  Currently defined printer types are:
[this list has been amended to support only the printer types that
are in current use today]

  epson
  Any Epson-compatible dot-matrix printer (default).

  hpdefault
  Any HP-compatible PCL printer.

  postscript
  Any Postscript-compatible printer.
  [this printer was not originally defined by the MS-DOS Graphics
  command, but we are adding it because many users have Postscript
  printers.]

/R
```

```
Print the image reversed (white on black).  Default is black on
white.

/B
Print the background in color, if possible.

/PRINTBOX:STD | /PRINTBOX:LCD
Print using the standard aspect ratio, or using an LCD aspect ratio
(1:1).

/LCD
Same as /PRINTBOX:LCD
```

help

Provides on-line help.

```
help [topic]
```

join

Join a drive to a directory.

```
join d: d:path
join d: /D
```

Options:

```
d:
The drive letter you want to set up.

d:path
The full path which will be assigned to the drive.

/D
Delete this join definition.
```

keyb

Configure the keyboard for a specific language.

```
keyb [xx[,nnn[,[d:][path]file]]] [options]
```

Options:

```
xx
The keyboard code.

nnn
The code page.

d:path\file
The full path to a keyboard definition file.  Default=KEYBOARD.SYS

/E
Use an enhanced keyboard.

/ID:nnn
Specify the keyboard for countries that have more than one keyboard
layout for the same language.
```

label

Assign a disk label to a drive.

```
label [d:] [label]
```

Options:

```
d:
The drive to assign a label.  If missing, use the current drive.

label
The label to assign to the drive.  If missing, prompt for it.
```

loadhigh, lh

Load a program into high memory.

```
loadhigh [d:][path]file [options]
lh [d:][path]file [options]
```

Options:

```
d:path\file
The program to load into high memory.
```

```
options
These are the options to the program that you are loading.
```

mem

Display the amount of memory installed, and the amount available.

```
mem [options]
```

Options:

```
/PROGRAM | /P
Display the programs that are loaded in memory.

/DEBUG | /D
Prints lots of debugging information.

/CLASSIFY | /C
Display the programs that are loaded in memory, and how much
conventional, expanded, and extended memory each is using.
```

mirror

Mirror information about the disk in a way that unformat and undelete can use to recover the disk.

```
mirror d: [/L] [/Td:entries] [/Td:entries] […]
mirror [/U]
mirror [/PARTN]
```

Options:

```
d:
The drive that you want to be able to recover later using undelete
or unformat.

/L
Retain only the latest information.

/Td:[entries]
Track information using a log file on the specified disk, and the
number of entries in the log file.

/U
```

```
Unload a previously loaded copy of the program.

/PARTN
Track information about the disk partitions.
```

mkdir, md

Creates a directory.

```
mkdir [d:]path
md [d:]path
```

mode

Set or display the operating mode of system devices.

Display the mode or status:

```
mode [device] [/STATUS]
```

If no devices are named, or if /STATUS is the only argument, display the operating mode of all devices. If a device is named with no settings, or with the /STATUS option, display the operating mode for this device. Set the mode:

```
mode LPTn: [options]
mode COMn: [options]
mode device codepage [options]
mode adapter [options]
mode CON: [options]
```

more

Displays a text file one screen at a time.

```
more < [d:][path]file
command | more
```

Options:

```
d:path\file
A text file that you want to display.

command
A command whose output you will pipe to the more program.
```

nlsfunc

Adds NLS (national language support) functionality.

```
nlsfunc [[d:][path]file]
```

Options:

```
d:path\file
The full path to a file containing NLS information.  If no file is
given, nlsfunc will select its own default.
```

pause

Suspend execution of whatever you are doing, and wait for a keystroke.

```
pause
```

print

Print a file in the background, while you run other DOS commands.

```
print [options] [[d:][path]file] [[d:][path]file] […]
```

Options:

```
d:path\file
The file that you want to print.

/D:device
Specify the name of the device to print to.  If not given, Print
will use the default LPT: device.

/B:size
Set the size of the buffer, in bytes, in the range 512-16384.  The
default is 512.

/U:ticks
The number of clock ticks, in the range 1-255, that Print is to wait
for the printer to become available before printing the job.

/M:ticks
The maximum number of ticks for sending a single character to the
printer.  In the range 1-255.
```

```
/S:ticks
The number of clock ticks, in the range 1-255, for the background
spooler process.  Default=8.  Larger numbers will speed up
printing but slow down other DOS programs.

/Q:nn
The number of files that will be allowed in the print queue, in the
range 1-32.  Default=10.

/T
Remove all files in the print queue.  Terminate the job that is
currently being printed.

/C
Cancel jobs in the print queue.  If this is used with a list of file
names, remove only those files from the queue.  If this is used
alone, cancel all jobs but do not terminate the current job.

/P
Add a file to the print queue.  If this precedes a list of files,
print only those files.  You may use /P and /C together on the same
command line.

If no options are given to Print, only files, assume /P.  If no
options and no files are provided, display the contents of the print
queue.
```

recover

Recover data from a bad diskette or hard disk.

[Dropped, because there are a variety of shareware and freeware disk recovery programs already avaialble, and it would be silly of us to write another one just because MS-DOS included a Recover command. If you need a disk recovery program, go download one, or purchase a third-party recovery program such as Norton’s Utilities.]

```
recover [d:][path]file
recover d:
```

Options:

```
d:path\file
The exact file that you want to recover, using file recovery.  Use
this when you know exactly the name of the file that needs to be
recovered.

d:
The drive letter of a diskette or hard disk to recover.  This
attempts recovery on all files.
```

rename, ren

Re-name a file or set of files.

```
rename [d:][path]file1 file2
ren [d:][path]file1 file2
```

Options:

```
d:path\file1
The source file, the original file that needs to be renamed.

file2
The new name for file1.  You may not include a path.

You can use wildcards (* and ?) to specify a set of files.  If you
use wildcards in file1, you must use wildcards in the same position
in file2.
```

rmdir, rd

Remove a directory.

```
rmdir [d:]path
rd [d:]path
```

scandisk

Scan a drive for errors, and report/repair any that were found.

setver

Set the DOS version that is reported to programs.

[Dropped, because this was a legacy migration thing, to allow programs that needed to run on a particular DOS version to be carried forward. Since later versions of DOS are (always?) compatible with earlier versions, a Setver command is not really needed.]

```
setver [d:][path] [file nn.xx]
setver [d:][path] [file [options]]
```

Options:

```
d:path
The path to the SETVER program data.

file
The name of a program that you want to add to the version table.

nn.xx
The DOS version number that will be reported to the program.

/DELETE
Remove this program from the version table.

/QUIET
Do not display any text while adding/deleting table entries.

If no options are given to SETVER, print the current version table.
If only ‘file’ is given, with no options, then print the DOS version
that will be reported to that program.
```

share

Share large files with other programs.

```
share [options]
```

Options:

```
/F:size
Set the size, in bytes, for file sharing information.  Default=2048.

/L:nn
Set the number of files that can be locked at once.  Default=20.
```

sort

Sort a file, or sort its input.

```
sort [options] [d:][path]file
command | sort [options] [d:][path]file
```

Options:

```
/R
Sort in reverse order.

/+n
Begin sorting at column n in each line of text.  Default=1.
```

subst

Substitute a path with a drive letter.

```
subst [d: [d:]path]
subst d: /D
```

Options:

```
/D
Delete this substituted drive.
```

sys

Make a floppy disk or hard disk bootable.

```
sys [[d:]path] d:
```

Options:

```
d:path
The (optional) location of the system files.

d:
The drive to make bootable.
```

time

Set or display the system's idea of the time.

```
time
```

```
time [hh:mm[:ss[.ss]]A]
```

Options:

```
hh:mm:ss.ss
The exact time to set for your system.  'hh' is the hour on a 12 or
24 hour clock.  'mm' is the minutes.  'ss.ss' is the seconds and
decimal seconds.

A | P
Set the time to AM or PM if using a 12 hour clock.

If no time is given, display the current time, and prompt for a new
time.
```

tree

Display the structure of a directory tree.

```
tree [d:][path] [options]
```

Options:

```
d:path
The drive and path to start from.  If no path is given, start from
the current directory.

/F
Display the names of files in each directory.

/A
Use ASCII characters to visually represent the directory structure.
```

type Display the contents of a text file.

```
type [d:][path]file
```

undelete

Recover a deleted file. Information from MIRROR may be useful.

```
undelete [[d:][path]file] [options]
```

Options:

```
d:path\file
The name of the file to recover.  If not specified, try to recover
all files in the working directory.

/LIST
List all the files that might be recoverable but do not recover
them.

/ALL
Recover all deleted files in the working directory.

/DOS
Recover only those files listed as deleted from DOS.

/DT
Recover only those files that are delete-tracked from the DOS
MIRROR mirror program.
```

unformat

Attempt to unformat a disk. Information from MIRROR may be useful.

```
unformat [d:] [options]
```

Options:

```
d:
The drive to attempt recovery.

/J
Checks that data from MIRROR is present and matches the
disk information.

/U
Attempts to unformat a disk without a MIRROR file.

/L
List every file and directory that might be recovered.

/TEST
```

```
Simulate a recovery, but do not attempt to recover the disk.

/P
Print the output on LPT1:

/PARTN
Attempt to recover the partition table.
```

ver

Display the version of the DOS kernel that you are using.

```
ver
```

verify

Turn file verification on or off. This tells DOS to check that files are correctly written to disk.

```
verify [ON | OFF]
```

Options:

```
ON | OFF
Turn file verification on or off.

If no options are given, display the status of file verification.
```

vol

Display the volume label of a disk.

```
vol [d:]
```

Options:

```
d:
The drive to display the volume label.
```

xcopy

Copies files and directories, including subdirectories.

```
copy d:  [d:][path] [options]
```

Options:

d:
The source to copy from. This must be either a drive or a full path.

d:path
The destination to copy to. If not present, assumes the working directory.

/A
Only copy files with the Archive bit set.

/M
Only copy files with the Archive bit set, and turn off the A bit.

/D:date
Only copy files modified after ‘date’.

/P
Prompt before copying a file.

/S
Copy subdirectories, except empty ones.

/E
Copy subdirectories, even if empty.

/V
Verify each file as it is written to disk.

/W
Wait before copying any files. Presents a prompt, which the user must first acknowledge.

The following commands are considered replacements for proprietary-named commands. These commands do not have the same name as their MS-DOS counterpart due to possible legal conflicts.

These may be implemented either internally or externally to the shell, and need not necessarily be identical to their MS-DOS counterpart. The following should be taken as guidelines and not a thorough design specification:

Alias

Create an alias for a command.

```
alias [aliascommand [realcommand]]
```

Options:

```
aliascommand
The new name for a command.  For example, 'ls' might be an alias for
the DIR command.

realcommand
The real command that is executed in place of realcommand, including
options.

If no options are given, display the list of currently defined
aliases.  If only the aliascommand is given, display its definition.
```

Backup And Restore

Commands to backup and restore a filesystem.

```
backup d:[path[files]] d:[path]
restore d:[path] d:[path[files]]
```

Options:

```
d:path\files
A list of files that need to be backed up.

d:path
The destination for the backup.  This may be a path or a drive
letter.
```

BASIC Language

Starts the BASIC language environment.

```
basic [[[d:]path]file]
```

Options:

```
d:path\file
The name of the BASIC program to load.  If no file is specified,
just open the BASIC environment.
```

Debugger

Perform debugging on a file, and perhaps some assembly.

```
debug [[[d:]path]file]
```

Options:

```
d:path\file
The name of a debugger script file.  The debugger should allow
the user to specify a file name to load by the script file.
```

Disk Defragmenter

Optimizes the hard disk space.

```
defrag [d:]
```

Options:

```
d:
The letter of the drive you want to optimize.  If you do not specify
a drive, it uses the working drive.
```

Line Editor

Allows a user to perform Stream Editing, either at the console or via a shell script.

```
edline [[[d:]path]file]
```

Options:

```
d:path\file
The name of a line editor script file.  The line editor should allow
the user to specify a file name to load by the script file.  Or, if
the script file does not specify a file name, use stdin and write to
stdout.  If no script file is specified, take input from the
console.
```

The following are exceptions, and must be provided for as internal commands in the shell:

call

Calls a batch file from within a batch file.

```
call [d:][path]file [options]
```

Options:

```
d:path\file
The name of the batch file to call.

options
The options that are passed to the batch file being loaded.
```

exit

Exits the shell.

```
exit
```

for

Runs a command for a each in a list of files.

```
for %var IN (list) DO command [options]
```

Options:

```
var
The name of an environment variable.  You will use this variable in
the ‘options portion.

list
A list of files that will be acted upon.

command
A command to run against the list.

options
The options passed to the ‘command’.  In the ‘options’, you may
reference the ‘var’ from above, preceding it with ‘%’.
```

goto

Branch to a label in a batch file.

```
goto label
```

Options:

```
label
The name of a label in a batch file.  The label must exist.  All
labels are defined by placing the label on a line by itself,
preceding it with ':'.  Spaces are not allowed in a label.
```

if

Perform a test in a batch file.

```
if [NOT] expr command [options]
```

Options:

```
NOT
Negates the expression.  If the 'expr' is true, NOT makes it false.

expr
A test expression, which evaluates to a true or false value.  Valid
expressions must include:
  ERRORLEVEL nnn
  Returns true if the error level is set to nnn.

  s1==s2
  Returns true if string 's1' equals string 's2'.

  EXIST [d:][path]file
  Returns true if d:path\file is a file.

command
The DOS command to execute if the test is true.
```

path

Set the search path for programs.

```
path d:]path[;[d:]path][…?
```

```
SET PATH=[d:]path[;[d:]path][…]
```

Options:

```
d:path
A directory that contains programs, to add to the PATH setting.

If no options are given to the PATH command, then display the
current value of the PATH setting.
```

prompt

Sets the prompt for the command shell.

```
prompt [text]
SET PROMPT=text
```

Options:

```
text
The text to use for the command shell prompt.  The '$' is a special
character, and can be used to insert special text in the prompt:
  $Q
  an equal sign

  $$
  a dollar sign

  $T
  the current time

  $D
  the current date

  $P
  the working path (drive and directory)

  $V
  the version of DOS (or the shell, either is acceptable)

  $N
```

```
the working drive letter

$G
a greater-than sign

$L
a less-than sign

$B
the pipe symbol

$_
a literal newline

$E
an escape character (can be used to start ANSI commands)

$H
a literal backspace character
```

```
If no options are given to PROMPT, display the current value of the
PROMPT setting.
```

rem

Ignores the current line. This is a comment.

set

Assign a value to an environment variable.

```
set var=value
```

Options:

```
var
The name of an environment variable.

value
The value to assign to the variable.
```

shift

Shift the position of all command line arguments downward by one.

```
shift [n]
```

Options:

```
n
The number of positions to shift the command line arguments.
Default=1.
```