INSTALLING THE DISK:

Assuming that your external drive is 'A', insert the disk and, at the DOS prompt type A:INSTALL. Directory \MAXNC will be created in your C drive with a sub directory named \E and the programs will be loaded automatically. The line: 'DEVICES=C:\DOS\ANSI. SYS' should be in the CONFIG.SYS file in the main directory for the screen plot program to work properly. The INSTALL program will add this line to the CONFIG.SYS if is not already there. it will save the original file under the name of CONFIG.XX1. Restart the computer, then type CD\MAXNC to change to the Maxnc directory. Work outside of windows, under the DOS command prompt.

SET UP:

Run program 'TIMESET' to calibrate the computer speed. This program will write initial values to the prmtrs.dat file. and set some of the parameters to match your computer speed The program will ask you for a factor number. Press enter to use the default for now. Wait for the program to finish (it should take less than a minute).

Type MAX to start the drive program and activate the machine (if CAPSLOCK is not on, press it). If not already done, plug MAXNC into the printer port and the wall outlet. The main menu should bean the screen. Test proper movement by activating the JOG mode (press 5 ) and pressing X. Using the shift keys to move the slide in the positive or negative direction. Check for smoothness. Do the same for the other slides.
OPERATION:

We'll start with an overview of the main menu. At the DOS prompt type MAX to enter the program and see the menu:

No. 1 ..LOAD PROGRAM is used to call up a NC program for MAXNC to execute. Answer with a path (if in a different directory), name and extension in DOS format.

No. 2 ..RUN PROGRAM will execute the NC program in memory. A new menu will appear for you to chose the mode of execution. During CQNTINUOS operation the space bar can be used to interrupt the program and hold movement of the slides (do not use pause).

No. 3 ..MDI manual data entry is used to manually command a move one line at a time and without a program.

No. 4 SEARCH is used to start the program at any line other than the beginning by searching for the specified string.

No. 5 ..JOG is used to manually move the axes, in fast or slow made.
No. 6 ..ZERO AXIS will zero the chosen axis register.

No.7 SCALE is used to change the scale of operation; normal is 1. 0000.

No.8 RESET is used to reset the program to the beginning and cancel any tool offset that may be active.

No. 9 ..OFFSET TABLE is used view and modify the tool length compensation table.

ESC .... To return to DOS. The posi5ons of the axes are saved on disk and wll be retrieved on MAX reentry.

Play around with all the menu options to familiarize yourself with all the screens. There are some sample NC programs that you can load and run to see the operation under automatic control, for example: press 1 and type 'MAXNC\WHEEL.T'. Jog to the center of the X and Y axes, bring down the Z axis to touch the material, (a piece of wood or aluminum 2" by 2" damped in a small vise will work) and zero all axes. press 2 and then 'C' far continuos (use a 1/16 end mill for tool) and watch a small wheel being machined.

PROGRAMMING

To write or modify an NC program, you'll need to use an editor such as EDIT in DOS, NOTEPAD in Windows or a word processor in document mode. The basic format of an NC program is based in the Cartesian coordinates to store the positions that the tool will move to, together with other information needed such as linear or circular move, feed rate etc. A single letter followed by a number are the basis of every command. The codes supported by Maxnc at this time are shown next:
G00 ...Rapid move  
G01 ...Feed controlled move  
G02 ..Clockwise are  
G03 ...Counterclockwise arc  
G12 ...Full circle clockwise  
G13 ...Full circle clockwise  
G43 ..Activate a tool offset  
G49 ...Deactivate tool offsets  
G61 ...Switch detect command  
G81 to G86 ..Drilling cycles  
G90 ... Absolute mode  
G91 ...Incremental mode  
G92 ...Absolute preset 

Fxx ...feed rate (speed)  
Hxx ...Tool offset to be activated 

Xx.x ..X coordinate  
Yx.x ..Y coordinate  

Zx.x ..Z coordinate  
Rx.x ..Drill rapid move  

I x.x ...Distance in X to center of are  
J x.x ... Distance in Y to center of are 

M30 ...End of program  
MOO ...Program stop  

M01 ...Optional stop  
M99 ...Repeat program 

These are the most common ones in the industry. More "G' and M codes will be added in the future and software upgrades will be made available. In addition, the following M codes are supported in the program but not in the hardware: M3, M4, M5, M7, M8, M9. They act on a second parallel port and can be utilized for programmed outputs. Their functions are explained in a separate sheet. 

Let's look at a short program and analyze the commands: 

G00X0.5Y0.5  ( rapid move to the x-y position indicated) 
G00Z0.1  ( bring the tool to the height indicated) 
G01Z-0.250F5.0  ( feed the tool down to the depth indicated) 
Y0.0  (linear move in Y at feed rate ) 
G02X-0.5YO.0I-0.5JO.0  ( make 1/2 a circle at same feed ) 
G01Y0.5  (linear move in Y at same feed ) 
G00ZO. 1  ( rapid move up in Z )
G81X0.Y0.OR0.02Z-0.30F2.0
   (drill cycle; rapid move in x-y then rapid to R height, drill to Z depth at feed and rapid back to R).
X0.312Y0.312  ( rapid to x-y and drill again as before )
X-0.312       ( same )
G80           ( cancel drilling cycle and return to Z0.1 )
GOOX1.0Y1.0Z1.0 ( rapid move in three axes simultaneously )
M30           ( end of program, move pointer to beginning )

Study the sample programs in the disk while operating the machine in single step mode and observing the movement of the axes (the last line on the screen is the one ready to be acted on when you press -S-). Most were designed with XO and YO in the center and the tool touching the surface for ZO.

CNC programming is not difficult to learn and the sample programs can help you get started, but the machine is capable of very complex operations that requires a deep knowledge of this language to take full advantage of that capability. Here are some suggestions on books that deal with the subject:

CNC A FIRST LOOK PRIMER by William W. Luggen
FUNDAMENTALS OF NUMERICAL CONTROL by William W. Luggen
COMPUTER CONTROL SIMPLIFIED by Michael Fitzpatric
COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS by G.E. Thayer

A lot of CNC programming today is done with the help of computers running CAD/CAM programs. These programs are usually very costly (in the thousands of dollars) but are a great help to the programmer. In order to provide our customers with at least some tools to create programs from CAD, we provide software to convert DXF files created from a CAD system into ready to run CNC programs.

THE CNC PROGRAM GENERATORS

The Autacad NC program generators are written in Basic and require that the Interpreter be in the path line directory for DOS to find it. The first step is to create a drawing in Autocad, then create POLYLINES for your tool paths (only polylines and points will be recognized by the generators). Create a DXF file (DXFOUT) and pick the polyline that represent the tool path The name must include the MAXNC directory as follows:

'DXFOUT' (enter) (file name?) \MAXNC\HOOK.DXF
(number of decimal places; E entities; ) E (enter) (enter), (enter).

Exit Autocad, go to MAXNC directory and type:
BASICA MILL or GWBASIC MILL or QBASIC /RUN MILL depending on your version of basic.

Use your editor to do any editing or to merge several programs into one.
For drilling or full circle operations, pick POINTS in the drawing and use DRILL instead of MILL to create the program.

Batch programs are available to simplify running these Basic programs but they work only with QBASIC. You would need to change them if you have any of the other interpreters. Their names are: MILL.BAT, DRILL.BAT and E.BAT. The "DRILL.BAS program can be used without a cad system by just entering the X-Y coordinates of the holes to be drilled.

PLOT

To view the tool paths on the screen, on the DOS prompt type PLOT or PLOT30 followed by a space and the name of the NC program you want to view (example: PLOT MAXNC\WHEEL.T" or for a 3d view type "PLOT3D MAXNC\CAR.I"). These programs operates in a similar way as MAX except that is necessary to press Min order to see the menu. There is also a set of parameters that may need to be modified depending on the type of graphics display adapter in your system. Use the PLOTPR program to change them; the values provided are for a VGA 640x480 video mode 18.

ENGRAVING

The engraving program is also a BASIC program and requires the interpreter to run. Type E and follow directions. The resulting NC program can be merged with others and can be viewed with the PLOT program. The individual characters are in a sub directory named K\ and are in the form of small NC programs of a 1” size. Only capital letters are provided. You can create your own fonts by making an NC program for every letter and saving it in disk with the prefix MTS for upper case or NTS for lower case.

LIMIT SWITCH INPUT:

The basic MAXNC-10 is not equipped with limit switches, however the software is set up to support limit switch inputs for a later implementation. The connections for the limit switches are to pins 10 and 12 of the parallel printer port. The program will run normally when these pins are in the low or grounded condition. A high or open <+5 volts >condition will stop movement after deceleration is completed. Pin 12 is used for limit switches in the positive direction for all axes. Pin 10 is for switches in the negative direction.

Note: Pins 10 and 12 will go high when open in most printer cards, but not all. Some cards require these pins to be damped high with a resistor for the limit switches to work.

SWITCH DETECT COMMAND (G61):

G61 is used to interrupt a move when a change of state happens in a switch connected to PIN 13 of the parallel port. Typical use is as a HOME position locator or PROBE sensor. Example: “G61X10.0” will move towards the X10.0 position until it reaches it, or the switch changes state. At this point it will continue with the next line on the program. This input is wired to the female receptacle on the right side of the machine. A digitizing probe and software is available for applications such as saving a contour 3 dimensional shape in disk.
<table>
<thead>
<tr>
<th>EXECUTABLE PRG</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAX.EXE</td>
<td>Main drive program</td>
</tr>
<tr>
<td>PLOT.EXE</td>
<td>Screen plot in 20 Used with NC programs no larger than 40,000 bytes.</td>
</tr>
<tr>
<td>PLOT3 D.EXE</td>
<td>Screen plot in 3D. Can be used with NC files as large as 400,000 bytes.</td>
</tr>
<tr>
<td>PR.EXE</td>
<td>Parameter setting program. Used to set-up and modify settings for the drive program.</td>
</tr>
<tr>
<td>PLQTPR.EXE</td>
<td>Same as above except is used on the Plots programs.</td>
</tr>
<tr>
<td>TIMESET.EXE</td>
<td>Program to calibrate the timing of the driver program to the speed of the computer. Needs to run before any other at set-up time and any time the Maxnc software is transferred to another PC. It is also used to adjust the speed of the system. For example if it is running too fast for the stepping motors to respond, then the factor in the program needs to be changed from the default value to a higher value (like from 1.2 to 1.5).</td>
</tr>
<tr>
<td>SEQSET.EXE</td>
<td>To set the proper sequence for the outputs. It modifies the &quot;ACODE.DAT&quot; data file to set the pins on the printer port to the appropriate values. It was set at the factory prior to shipping. It is needed only if the wiring of the stepping motors is changed.</td>
</tr>
<tr>
<td>ACODE.EXE</td>
<td>To set the sequence manually. Not needed for normal operation.</td>
</tr>
</tbody>
</table>
Using the PR.EXE program to change parameters.

The PR.EXE program allows you to modify the PRMTRS.DAT file. Here is an explanation of these parameters:

1- JUMP START SPEED. This is the jump or initial speed for the motors. The motors will accelerate from this speed if the program feed rate will allow it. Increase to slow down.

2- MAXIMUM SPEED. Maximum speed in RAPID mode (G00) and in FEED mode (G01,G02,G03 etc.). Increase to slow down.

3- SLOW JOG. Controls the speed of the SLOW JOG MODE. Increase to slow down.

4- ACCEL. FACTOR Controls the degree to which acceleration occurs between jump speed and maximum speed. Increase to accelerate faster.

Parameters 1 to 4 and parameter 8 are modified by the TIMESET program. If you change them, don't run TIMESET again unless you want to reset them.

5- X BACKLASH Play in the drive can be compensated with this parameter. It reads in steps. To convert, multiply inches by parameter #8 (ea. 0.0023" x 8000 = 18 steps).

6- Y BACKLASH Same as above but for Y axis.

7- Z BACKLASH Same as above but for Z axis.

8- FEED CONTROL Affects the proportion of the stated program feed rate. Change this if the programmed feed rate is not accurate (in inches per minute).

9- X RESOLUTION Sets the resolution for this axis. Is given in STEPS per INCH (or STEPS per CENTIMETER for metric systems). To arrive at this number, multiply the STEPS per revolution of the motor by the revolutions per INCH of the drive (or screw).

10- Y RESOLUTION Same as above but for the Y axis.

11- Z RESOLUTION Same as above but for the Z axis.
12-PORT ADDRESS 1 | X and Y codes are output to this address. Is a decimal number, normally 888. It will have to be changed only if your computer is not totally IBM compatible, or the printer port card is not set to the primary address.
---|---
13- PORT ADDRESS 2 | Z codes are output to this address. Normally 890. It will have to be changed only if your computer is not totally IBM compatible, or the printer port card is not set to the primary address.
14- PORT ADDRESS 3 | Input signal come from this address. Normally 889. It will have to be changed only if your computer is not totally IBM compatible, or the printer port card is not set to the primary address.
15- PORT ADDRESS 4 | For future expansion (4th. axis et.).
16- PORT ADDRESS 5 | Outputs for "M3*, "M4", "M5", “MT", "M8" and "M9". They are active in the program but not implemented in Maxnc-10.
17- PORT ADDRESS 6 | For future expansion
18- PROMPT FOR TOOL CHANGE | Setting this parameter to other than 0 will interrupt NC program execution on any "M6" in the program and prompt you for a tool change.
19- STOP FOR OPTIONAL STOP | Setting this parameter to other than 0 will interrupt NC program execution on any "M1" in the program.
20- RESPOND TO LIMIT SWITCH | Setting this parameter to other than 0 will disable limit switch sensing. It needs to be set at 1 for Maxnc 10 (unless limit switches are installed by the customer).
CHECKLIST.

1...PLACE THE “MAXNC” DISK IN THE FLOPPY DRIVE AND TYPE “A:INSTALL “. The program will copy the line "DEVICE=C:\DOS\ANSI.SYS" to the CONFIG.SYS .(the file "ANSI.SYS must exist in the \DOS directory). AFTER IT FINISHES, RESTART THE COMPUTER.

2...FROM THE ‘\MAXNC\’ DIRECTORY (in DOS), RUN THE PROGRAM "TIMESET" TO CALIBRATE THE PARAMETERS TO THE SPEED OF YOUR COMPUTER.

3... RUN THE MAIN PROGRAM "MAX" AND TRY THE MODE JOG (5) . RUN DIFFERENT AXES AND CHECK FOR FREEDOM OF MOVEMENT. TRY ALL THE COMMANDS AND SOME OF THE NC PROGRAMS PROVIDED,

IF YOU NEED TO MAKE ADJUSTMENTS IN SPEED, RUN THE TIMESET PROGRAM AGAIN OR USE THE "PR" PROGRAM TO CHANGE THE PARAMETERS INDIVIDUALLY.

‘M’ codes for a secondary parallel port

M3.....Turns pin # 1 on. Commonly used to start the spindle clockwise.
M4.....Turns pin # 14 on. Commonly used to start the spindle counterclockwise.
M5.....Turns pins 1 and 14 off.
M7.....Turns pin 16 on. Commonly used to turn spray coolant on.
M8.....Turns pin 17 on. Commonly used to turn flood coolant on.
M9 ....Turns pins 16 and 17 off
The 4th. axis Adapter for the MAXNC-10 CNC Machining Center expands the capabilities of the Machine with the ability to rotate the part being machined, under program control, in full interpolation with the other three axis.

The adapter includes a 4.0" diameter rotary table, which is normally mounted with the axis of rotation perpendicular to the spindle of the machine and parallel to the X axis.

The 4th. axis is furnished with its own control box that includes its own power supply. As a bonus, the control provides connections for the spindle motor so that spindle start and stop can be controlled from the program.

The resolution of the system is 0.0125 of one degree per step.

INSTALLING THE PROGRAM:

After inserting the disk in the "A" drive, type: A:INSTALL

The software will be copied into the MAXNC directory. If the directory has not been created in the hard drive of your computer, then you need to install the original MAXNC software first, then install the 4th. axis software second.

The 4th. axis driving program, called MAX4.EXE, looks the same as the 3 axis program except for the addition of the "A" axis register below the "Z" register.

All the "A" axis functions operate in the same way as they do in the other axes. The register for the "A" axis is set to display to 3 decimal places. One revolution of the motor rotates the rotary table 5 degrees. Since there are 400 half steps per revolution on the motor, the resolution of the 4th. axis is 0.0125 degrees.

Rotation is clockwise for a positive move commended in the program, or by manual jog

The faceplate of the rotary table is constructed with 4 "T" slots for mounting parts, fixtures, chucks etc.
MOUNTING THE 4TH. AXIS ON THE MILL:

Clamp the rotary axis on the left side of the table of the MAXNC 10, using 4 "T" nuts, screws and hex nuts. Before tightening the nuts, indicate the faceplate of the 4th. axis to be parallel to the "Y" axis. Note: the rotary table is relatively heavy. This helps to dampen vibration very nicely but it also places a heavy burden on the "X" axis motor. Care should be taken that the rotary table is not operated too far to the left without some counterbalance on the right.

The 4th. axis control box can be placed on the left side of the MAXNC 10 control box with the air intake facing to the rear. Make sure that the air intake is not obstructed and that it is free from taking in any chips from the machine. Connect the stepper motor cable to the connector of the 4th. axis control box.

If you want your programs to turn the spindle on and off automatically, disconnect the motor from the MAXNC 10 control box, then connect it to the cable coming from the 4th. axis box, then this cable into the MAXNC 10 control box.

The 25 pin port cable connects to a secondary parallel parton the PC.

PROGRAMMING:

Programming the 4th. axis is done with the "A" word. For example, a rapid move to an absolute position of 90.000 degrees is done as follows:
GOOA90.0
A command to move all 4 axes simultaneously at a specified feed rate would look like this:
G01X1.5000Y2.000Z0.2000A-180.0F30.

Controlling the spindle is done with "M" commands: "M3" turns the spindle on; "M5" turns the spindle off. "M30" (Program reset) will also turn the spindle off, as well as resetting from the main menu (8).

USING THE MILL4X.EXE PROGRAM;

The MILL4X.EXE program can be used to create rotary axis programs, automatically, from flat CAD drawing DXF files. Draw the tool paths in POLYLINES, or convert LINES and ARCS into POLYLINES, in the same way that you do with the three axis program generator.
Export the drawing to a DXF file. Run the MILL4X.EXE program with the name of the DXF file as follows.
"MILL4X dxfname.DXF"

dxfname is the name you gave the DXF file when you created it in the CAD system. If the file is in a different directory, type the whole path name. Don't forget the space after MILL4X. If you are working in Windows, just drag the DXF file to the MILL4X.EXE program to execute.
The "Y" axis values in the CAD drawing are converted to "A" values, after adjusting them for the diameter of the part.

If you need to machine a series of features around a cylinder, and you need to place them equally spaced, then draw the tool path (with POLYLINES) in your CAD system, then use COPY or ARRAY, separating the paths in the "Y" axis by an amount equal to the following formula:

\[ Y = \text{DIA} \times \pi / \text{SPACES}. \]

Example: six pockets need to be milled around a 1 5 Dia. bar. Draw one pocket at XO and YO then copy it 5 times along the "Y" axis a distance of:

\[(1.5" \times 3.14159 / 6) = 0.785397" \text{ each.}\]

When You run the DXF file with the MILL4X.EXE program, enter the same diameter that you use in the formula and the program will replace the "Y" linear values with the proper angular values. Since there is no circular interpolation in a rotary axis, polyline arcs are converted to a series of straight lines to simulate a curve around the diameter.

Some sample NC programs are included to demonstrate the process of 4th axis programming, including the DWG. files (from AUTO-CAD), the DXF files, and the CNC programs (with the "T" extension).