



Numatix Control Panel

In addition to conventional CNC functions, the Numatix CNC Control performs the following:

- High speed, high accuracy machining - 80 smart look ahead buffers (about 80 mm or 3 inches)
- Eight axis linear, circular and helical simultaneous interpolation
- Polar and spherical coordinates, azimuth-elevation / pitch-yaw and roll
- Look-ahead cutter radius compensation, chamfering and automatic rounding of any corner
- User defined canned cycles, G and M codes
- Subroutines, functions, loops, chaining and calling programs
- Built-in menu generator and data entry to create conversational programs
- Color graphic display, powerful multi-window editor

- Common computer chores: number crunching, text and file processing, screen manipulation
- PLC, implemented in software, runs in the background

Specialty Features

On The Fly (Dynamic) Z Axis Offset

The one great feature about a knee mill is the ability to crank the table up or down as desired even while cutting the part. This is now possible in the Numatix control, and can be utilized in all axes, not just in Z.

Each axis can be cranked into a new relative position even while cutting your part. The offset can be changed, by the handwheel and the jogging pushbuttons, before or while the program is running.

2 Axis Programming from CAD is Automatically Wrapped Onto a 4th Axis Cylinder

Developed especially for the roller die industry, this feature saves many hours of CAD time.

Setup Tool Length

A pop up window lets the operator enter tool length either as a number (if known) or as measured on the machine. The operator can use or ignore tool radius (for ball nose) and enter the current Z, if not zero, which is saved for the next tool.

In addition, tool length can be incremented.

Sequence Display

Any automatic sequence that may require operator intervention, e.g. tool changer, can be displayed in real time. The full sequence is displayed while a moving color bar highlights the current operation. If an error occurs the erroneous operation stops and blinks. The operator can go backward and forward by the arrow keys. Once the problem is solved the program can be resumed.

Custom Menus

Your control comes complete with all the tools available to customize your screen. Many customers find it valuable to generate custom menus that prompt the operator through their part setup or to complete complex formulas based on the operator's input.

Run Simultaneous Programs

It may be advantageous to run other programs on your control in between CNC operations. With Numatix you are able to call up a SPC (or other) program, record key information, then resume your CNC program without rebooting, re-homing, or searching for the point where you stopped in the program.

Powerful Editor

Powerful word-processor for interactive program editing:

- Up to 250 MB or 1,000,000 lines file size
- Up to 5 simultaneous open windows, with zoom
- Automatic generation of backup file
- Automatic program save before exiting the editor
- Moving and copying text inside a window or between windows
- Search, search and replace, compare two files
- Merging or splitting of programs
- Syntax check for the program; if error is detected the cursor points to the erroneous data
- Point and shoot directory and file selection
- 10 bookmarks can be set and recalled
- Undo changes to a line, undelete lines
- Print to a printer

Mid Program Restart

Your Choice! Restart from – any given line number – any given block number – a percentage of the program – or your tool's current location. The control will read all previous commands and begin work from the point you have defined in seconds. The position and spindle are resumed simply by pressing Cycle Start and following on-screen instructions.

Fully Independent Axis Control

Some controllers require identical mechanical and electronic feed back to be present before the control will function correctly. Your Numatix control will maintain the correct, coordinated feeds and speeds regardless of mechanical axis properties. A machine that has a X axis with an odd pulley ratio and 1000 lines encoder can work with coordinated precision when tied to another axis which has a direct drive and a 5000 lines encoder. Numatix allows you to mix torque mode control on one axis and velocity mode on another. Even with a 4th axis rotating part or a 5 axis contouring move the cutter motion is based on the actual speed of the cutter tip as it travels across the part.

Jerk Factor

Jerk is the rate at which the acceleration changes. Jerk factor is the time it takes the acceleration to rise and fall, and it converts step acceleration into a trapezoidal or triangular acceleration.

Two linear jerk factors can be set up to 250 msec- one for positioning and another for interpolation.

The Jerk factor generates bell-shaped feed profile and is essential for high-speed contouring.

Geometry Correction

The Numatix control has the internal power to correct for machine out of square conditions. Some times it is impossible or very costly to correct for poor machine geometry. The Numatix control provides internal formulas to electronically correct for machine geometry error. Let's take this one step further. Suppose the only way a part will fit onto your machine is by mounting it askew to the axis. You can now establish a new working plane which allows the use of your standard X, Y, Z, program. In this example the Numatix control will output coordinated motion when your program commands a straight line move in only one axis.

5 Axis Programming

Many 5 axis moves are really 2 or 3 axis moves in a rotated plane. With a Numatix control on your 5 axis machine tool, you can program in simple 3 axis mode after placing the control into a rotated and/or titled plane.

Security

If security is a factor, the Numatix CNC Control Software source code can be deleted from the control (the compiled file is unreadable and useless for competitors).

For greater security, up to 9 levels of access are provided via user name and password.

Any access is logged to a file.

More Standard Features...

Spindle/Axis

The spindle motor can be programmed both as an axis in degrees or as a spindle in RPM, as long as it has an encoder. This is used for some tool changers and for rigid tapping.

The mode can be switched anytime without losing position.

Tool Changer

All kinds of tool changers can be programmed in the internal PLC.

There are four basic types of tool changer:

- The tool is picked up solely by axis motion
- The tool is taken directly from the magazine, with or without axis motion
- The tool is exchanged with a new tool from the magazine by a Tool-Changer-Arm (TCA). This is also called

Random ATC because tools are not returned to the pots from which they were taken

- The TCA exchanges tools with an Intermediate Arm, which is loaded ahead of time from the magazine

For M6, T represents actual tool number. For a random ATC, the computer finds and updates the pot number in the tool table. For non-random ATC, the pot number is set to the tool number. M6 also sets the tool offset according to the entry in the tool table.

Endless Rotary Axis

When set, such an axis behaves as a linear axis for interpolation but its limits are set internally for thousands of revolutions.

Unwinding is performed automatically in the following cases:

- Jogging to zero or positioning command: The axis moves at the shortest path (180 deg or less)
- Incremental programming of zero move: There is no motion but the current position is reduced by multiples of 360 to the range +/-180 deg
- Spindle operation: When the axis is used as a spindle it is constantly kept in the +/-180 deg range

Follower/Axis

Any axis can be switched to Follower mode.

In this mode, the axis is moved by a feed command rather than position. A follower axis can follow feedback signal (force, position), pushbuttons, another axis, etc.

The mode can be switched anytime without losing position.

Differential Pair

Two axes can be setup as a differential pair, using the ratio between them.

This is required for some CA heads in which the A-axis is driven through the C-axis.

Gantry

Up to 2 gantry axes, each driven by 2 independent servos, are supported.

If necessary, the Reference routine can align the 2 servos to ensure squareness.

Ball and Bull Nose Cutter Radius Compensation

This compensation is used for 3D contouring. It requires a post-processor that can generate IJK vector in addition to XYZ.

It also works with 5-axis profiling.

Drilling Cycles

Standard drilling cycles work with 5-axis TCP.

Rigid tapping can be used on machines with servo spindle.

N Display

Three different displays can show where the part-program is. They can be switched any time:

- Block number (N), as appears in the program
- Sequential line count, regardless of N
- Percentage of the program in 4 digits

PLC

The PLC is implemented in software and runs constantly in the background.

It is used for background operations, such as preparing the next tool, spindle control, display, alarm, watchdog, etc. The PLC reads and writes I/O's from an I/O board with a supplied driver. The I/O ports are numbered by the retrofitter so there is no need to use absolute addresses. The PLC copies the I/O's to/from PLC designated registers. In addition, it intercepts all keyboard keys; these key-codes can be modified, used as inputs, or pass-thru.

700 registers are available for use. Of the 700 registers, 200 registers are associated with the control software variables. These 200 variables can be read and/or written, such as mode, feed, position, leadscrew compensation, trace-error, etc.

PLC programs are written and compiled using the control software editor, no additional tools are needed.

PLC registers can be displayed on the screen in real time in different formats. In addition to numerical values, they can point to text messages with variable colors.

More Standard Features

Note: the terms 'indefinitely' and 'unlimited' are used when a quantity is limited by memory size or when the limit is in the quadrillions and beyond any need.

Basic Features

- 8 axis positioning
- 8 axis linear interpolation
- 3 axis circular interpolation
- 2 or 3 axis circular and all other axes linear, simultaneous (helical) interpolation
- Up to 250 MB program size
- Up to 1,000,000 miles dimension for any axis or radius (one trillion mm)
- 80 look-ahead buffers for high speed cutting of cad/cam output
- 6 digit Spindle RPM
- 7 digit block number (Line count can be used instead of block number)
- Cartesian, polar, toroidal and spherical coordinates
- Rotation and elevation to any program, including cutter compensation
- Inclined polar coordinates (Tilt and swivel)
- Automatic rounding for 3D corner
- 3 point 3D arc
- 3 point 3D full circle
- Feed input
- g93 inverse time
- $\text{Feed}[\text{unit}/\text{min}] = \text{Length}[\text{unit}] * \text{Inverse_time}[1/\text{min}]$
- g94 unit/min
- g95 unit/rev
- Feed in ipm or mm/min for a rotary table
- Programmable scale for Feedrate
- ipm or mm/min independent of scale or inch/metric switching
- Feed profile
- Bell-shape (S curve)
- Programmable Jerk Factor
- Programmable automatic deceleration in corners
- Programmable feed override, up to 31 steps 0% to 199%
- Programmable G0 override 1% to 100%
- 9 settable fixture offsets, extendable to 99
- Mixed absolute and incremental input
- Programmable inch/metric, mixed in the same block
- Programmable zero offset and local coordinates
- Programmable scale-factor, independent for each axis
- Programmable mirror image for all axes in any combination
- Programmable plane for circular interpolation and radius-comp

- g17 to g19 for XYZ
- g16 for user defined axes
- Selected by 'Plane(axis1,axis2,axis3)' command.
- Dwell up to 300,000 years
- 15 digit data, written with floating decimal point. No leading or trailing zeros are necessary
- Follow the leader - any axis can lead any number of other axes, in any combination (For all kinds of coordinates and motions)
- Gearbox - any axis command can be multiplied by a gear ratio
- Arithmetic
- Direct programming of arithmetic expressions with any level of complexity, written in a subset of BASIC language
- NC addresses can be used on the right side of an equation
- File handling
- Text and random-access
- Open, close, read, print, seek, put, get, eof, lof
- DOS programs can be called from within a CNC program
- Text processing
- Input, modify, append, compare, examine, print
- Date and Time functions for scheduling maintenance
- Unlimited number of sub-programs
- Loops
- For ... To ... Next
- While ... Wend
- Repeat ... Until
- Loops may be nested indefinitely
- Unlimited number of repetitions of any loop
- Chain - a program can load and run another program
- Call - a program can call another program as a subroutine, up to 16 levels of nesting
- 100 general-purpose parameters, 15 digits each
- 100 special purpose parameters, for programmable canned cycles
- 8200 parameters, can be used as general purpose or arranged in user-defined vectors and matrices
- 700 PLC's 16 bit variables
- Support for Mouse, Touch-screen, RS-232
- Tool radius and length compensation
- Ball and bull nose 3D compensation
- Up to 253 tool data storage, arranged in a pop up table
- Tool data given in mixed inch/mm, independent of the program
- Mixed tool radius and tool diameter
- Tool data is called by tool number
- Tool data can be checked and modified by the program
- No-excuses, look-ahead cutter radius compensation
- Covers all transitions: line to line, arc to line, line to arc and arc to arc, external and internal corners
- Look ahead search goes as deep as necessary
- Up to 10 Z-blocks and M-functions can be properly handled by the look ahead
- Z motion inside corners is handled automatically
- Automatic rounding of corners by a programmed radius. All corners are covered:
- Line to line, arc to line, line to arc and arc to arc
- All cases of external and internal corners are handled automatically
- Automatic chamfering for any corner between two lines
- Chamfer may have rounded corners

- Internal chamfer is automatically ignored if the tool does not fit in
- Center-line programming for slots
- Meander milling with tool compensation
- Semi-automatic recovery from power loss
- Skip function enables interrupting a motion or dwell upon a programmable input condition
- Leadscrew compensation, dynamically interpolated
- Stored end-of-travel limits

Program Debugging Facilities

- Graphic
- Dry-run
- Test
- Single
- Break-points
- Messages

Program Display Exhibits The Following (can be user arranged)

- Current mode
- Program name
- Current & buffered program lines
- Axis position
- Block and tool number, actual feedrate and spindle-speed
- Time of date
- Error messages
- INCH/MM, Dry-run, Single, Home #, Feed-override
- Bottom line instant help
- Graphic projection and isometric view of tool path
- Diagnostic tests

Pop-ups

- Main menu
- Tool table
- Parameter table
- I/O status
- Help screens
- File directories
- Context sensitive help screens can be displayed any time. These help screens include all G-codes, the Letter-addresses, MDI commands, editing commands, and more
- Help text can be included in any CNC file

Miscellaneous

- Transformations (for 5 axis machines)
- World coordinates
- Wrist coordinates
- TCP
- Electronic roll
- Calculator mode may be used in MDI
- Graphic display
- Projection and isometric, make-BMP command
- Program may be started regardless of current axis position

- Dialogue windows
- Enables programming of menus and instructions to the operator
- Operator can enter data on line
- Programmable standard drilling, boring and tapping canned-cycles
- Tool radius compensation may be used with any selected plane
- With radius compensation, a few programming chores can be performed automatically:
- Avoiding deceleration when possible
- Chamfering
- Rounding of corners
- Finding intersections between a line and an arc
- Finding intersections between two arcs
- Centerline programming for slots
- Built-in diagnostic and calibration
- Trace error and time-constant
- Reference test (without jogging to the reference point)
- I/O status for all inputs and outputs
- Boot-log file for tracing boot sequence

Site Map

[Home Page](#)

[About Numatix](#)

[Why Numatix Controls!](#)

[Save Time, Increase Profit](#)

[Contact](#)

Calendar

March 2018

S M T W T F S

				1	2	3
4	5	6	7	8	9	10
11	12	13	14	15	16	17
18	19	20	21	22	23	24
25	26	27	28	29	30	31

Search Site