# CNC USB Controller

## User manual

## 2010-06-26

# Table of Contents

# 1 Introduction

## 1.1 Overview

CNC motion controller is a link between personal computer and drivers for stepper motors. It uses USB port which is available on all modern computers and laptops. This is a complete (software/hardware) solution and it does NOT require any additional software.
USB CNC controller is compatible with most step/dir drivers. It can be used as direct replacement for many parallel port break-out boards.

## 1.2 System Requirements

**Minimum system requirements:**
1 GHz or faster processor
512MB RAM
500 MB available hard disk space
DirectX 9 graphics device with WDDM 1.0 or higher driver
USB 2.0 port
.NET Framework 3.5 SP1

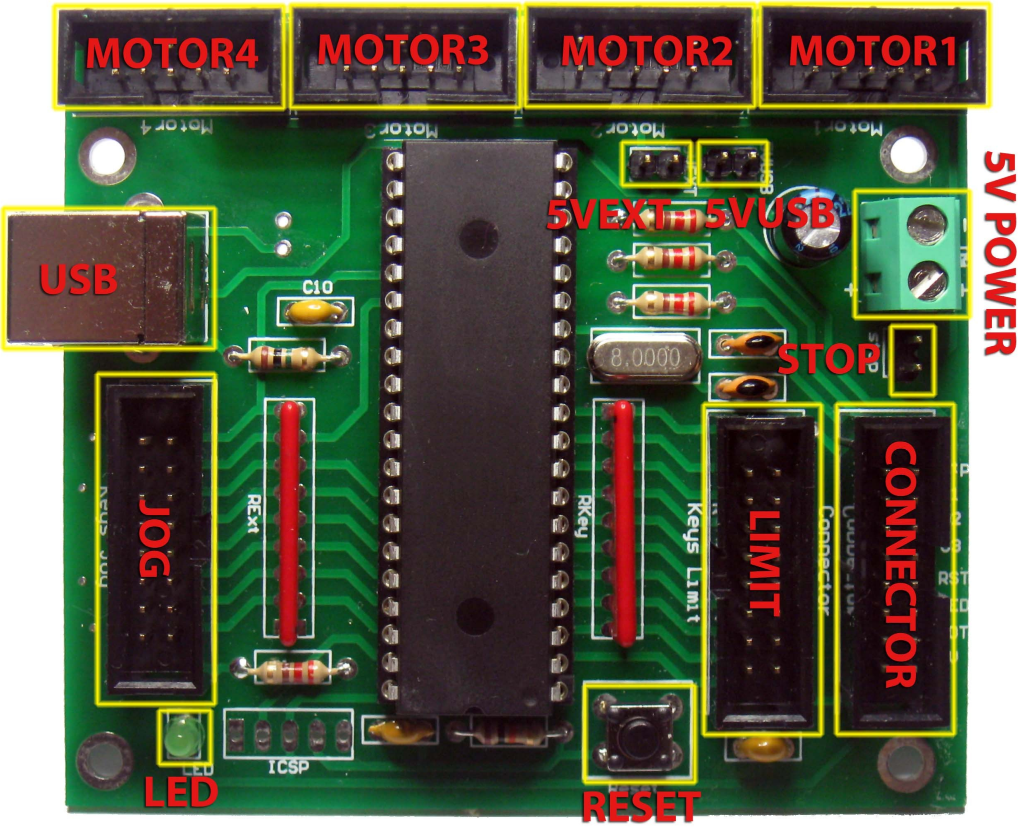
**Recommended system requirements:**
2 GHz or faster processor
2GB RAM
500 MB available hard disk space
DirectX 9 graphics device with WDDM 1.0 or higher driver
USB 2.0 port
.NET Framework 3.5 SP1

# 2 Hardware

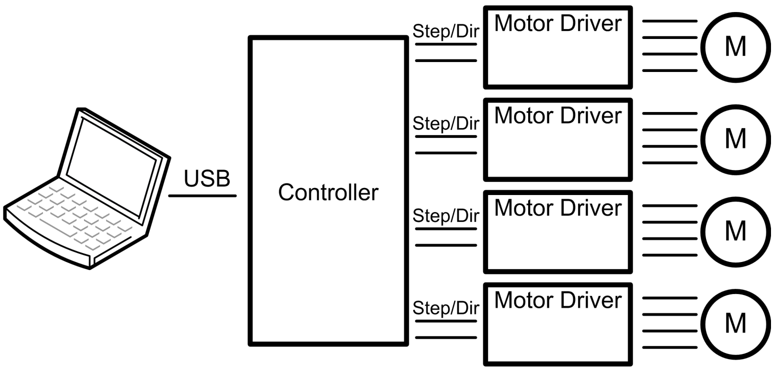
## 2.1 Specification and features

- 4 axes USB CNC controller
- USB (V2.x) from PC/Laptop
- motor driver connector pin-out is compatible with 10 pin open source interface
- controller works with most motor drivers available on the market
- 3 digital outputs (flood, mist, spindle)
- 25 kHz maximum step frequency
- 12 us minimum pulse width
- buffered IO for maximum performance
- advanced interpolation algorithms
- two manual jog input keys per axis (8 total)
- two limit keys per axis (8 total)
- pause/resume of execution supported
- automatic homing procedure
- advanced toolchange procedures
- standard RS274/NGC G-code (EMC2 compatible)
- toolpath simulation
- advanced G-codes - G40, G41, G42 (Cutter Radius Compensation) supported
- advanced G-codes - G43, G49 (Tool Length Offsets) supported
- advanced G-codes - G54, G59.3 (Coordinate System Origins) supported
- tested with SolidCAM, MasterCAM, ArtCAM, Vectric, ... generated G-code
- Profili 4-axes and 3-axes G-code supported
- import toolpath from DXF file
- import toolpath from PLT/HPGL file
- import toolpath from image file
- export toolpath to G-code
- export toolpath to DXF
- SDK (software developers kit) is available

## 2.2 Board description



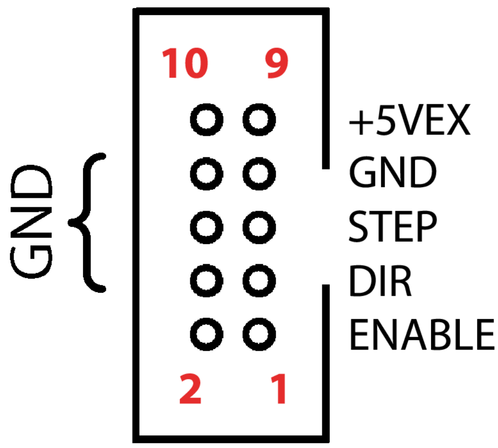
## 2.3 Installation




Controller must be connected to motor drivers which drive motors. Computer is connected to USB connector.

IMPORTANT: Do not connect controller to USB without supplying power. Either place 5VUSB jumper or an alternative power source. See power options described below.

### 2.3.1 *Motor* connector



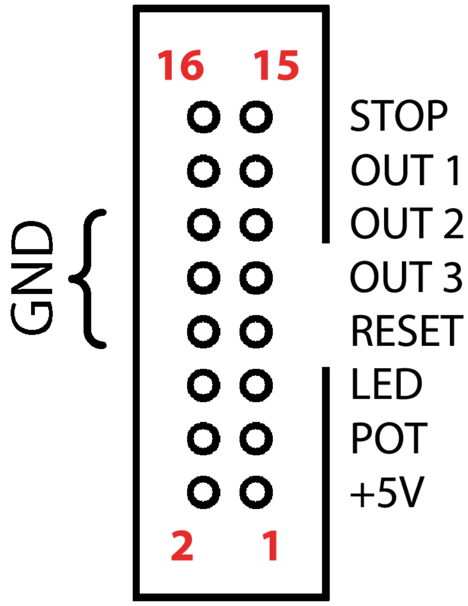

*+5VEX* – 5V power of motor driver. This pin can be used to power motor driver from controller or to power controller from motor driver. 5VEX jumper on controller must be closed to enable this connection.

*STEP* – Step signal for motor driver. Width of signal is 12uS or more.

*DIR* – Direction signal for motor driver.

*ENABLE* – Enable signal for motor driver. Controller does not control this pin. It just connect them together and provides possibility to easily connect emergency stop switch. You can connect emergency stop switch to to *stop* pin on *Connector* connector.

### 2.3.2 *Connector* connector




*STOP* – use this pin to connect emergency switch. This is same as *Stop* jumper.
See *Motor* connector ENABLE pin for more information.

*OUT1* – this pin is controlled with spindle commands

*OUT2* – this pin is controlled with flood coolant commands

*OUT3* – this pin is controlled with mist coolant commands

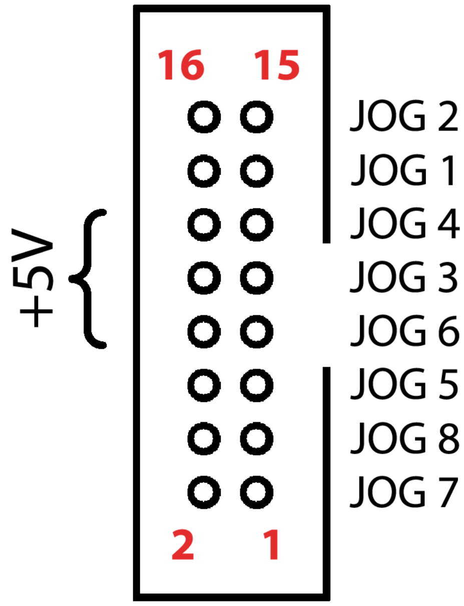
*RESET* – connect this pin to GND to reset controller (same as *Reset* button)

*LED* – use this pin to connect external blinking LED. You must use current limiting resistor to limit current to 10mA.

*POT* – connect 5k or 10k ohm, logaritmic taper potentiometer to this pin to controll speed of manual jogging.

+5V – 5V power supply for  potentiometer

### 2.3.3 Jog connector




*JOG 1-8* – for connecting jog keys. It is recommended that you put 100nF capacitor directly to key teminals.

Typical connection is:

JOG1 – X- axis

JOG2 – X+ axis

JOG3 – Y- axis

JOG4 – Y+ axis

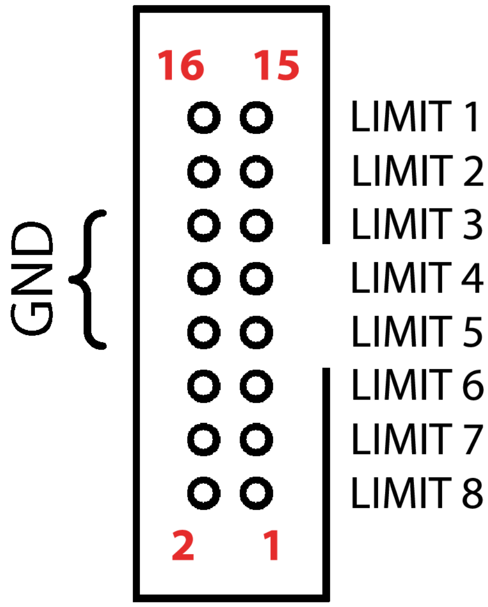
JOG5 – Z- axis

JOG6 – Z+ axis

JOG7 – A- axis

JOG8 – A+ axis

### 2.3.4 Limit connector




*LIMIT 1-8* – for connecting limit switches. It is recommended that you put 100nF capacitor directly to key teminals.

Typical connection is:

LIMIT1 – X- axis

LIMIT2 – X+ axis

LIMIT3 – Y- axis

LIMIT4 – Y+ axis

LIMIT5 – Z- axis

LIMIT6 – Z+ axis

LIMIT7 – A- axis

*LIMIT8 – A+ axis*

### 2.3.5 *Stop* jumper

Close this jumper if your motor drivers support ENABLE signal and you don't want to use it.

### 2.3.6 *5VUSB* jumper

Close this jumper to power controller from USB.

When closed, *5V power* connector (green) must not be connected.

When closed, *5VEXT* jumper must be open unless you want to power motor drivers through controller.

### 2.3.7 *5VEXT* jumper

Close this jumper to power controller from motor drivers or to power motor drivers.

Use only if you know what you're doing. Incorrect usage may damage controller and/or drivers.

### 2.3.8 *5V power* connector (green)

Use this connector to power controller with external 5V power supply.

**WARNING**: CHECK POLARITY AND VOLTAGE

Controller is not protected against wrong polarity or voltage.

When in use, *5VUSB* jumper must be open.

When in use, *5VEXT* jumper must be open unless you want to power motor drivers through controller.

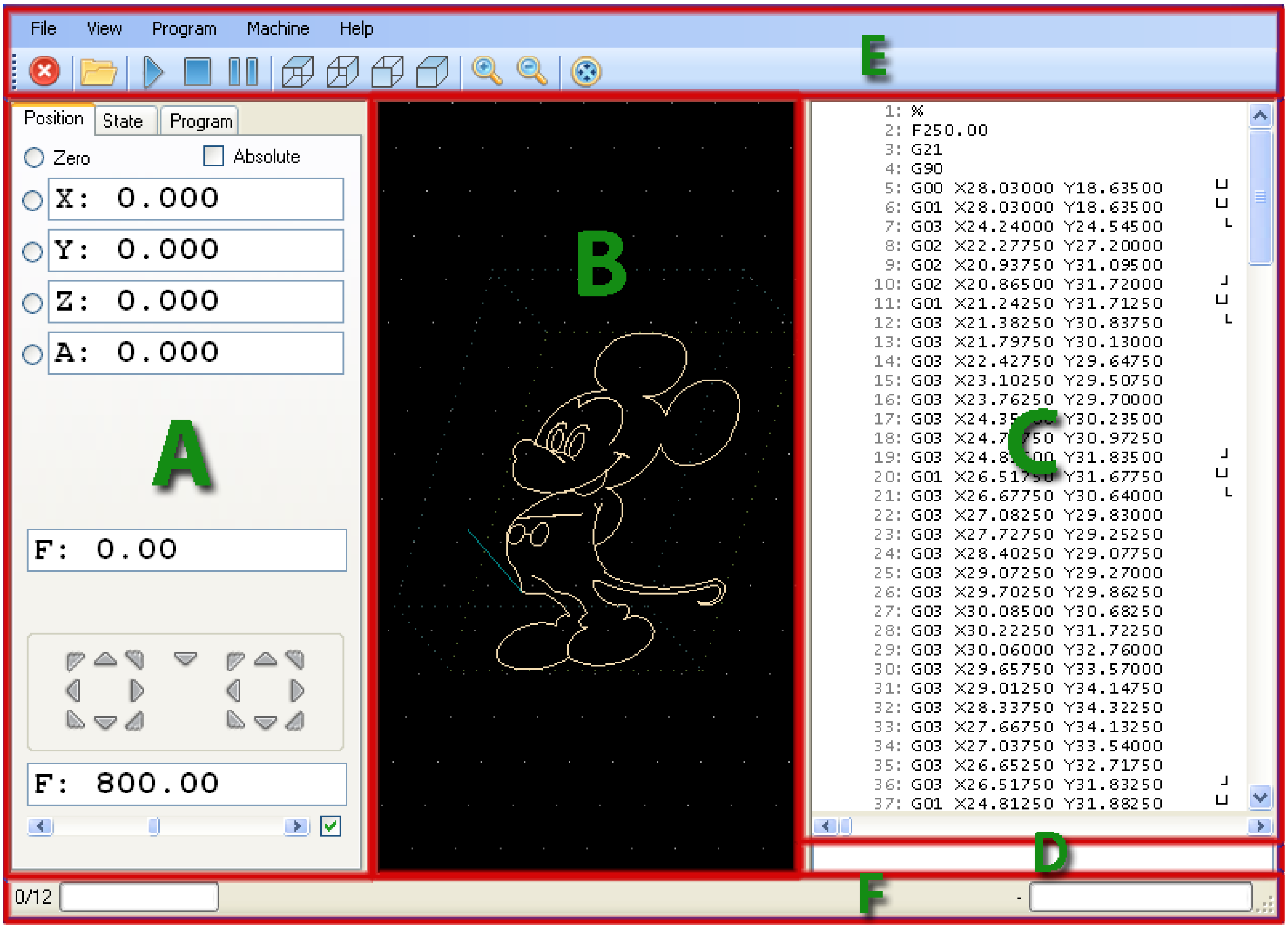
# 3 Software

## 3.1 Overview

CNC USB Controller software is specialy designed to work with controller. All controller functions are accesible through software.

Software is designed to be simple and friendly to new users.

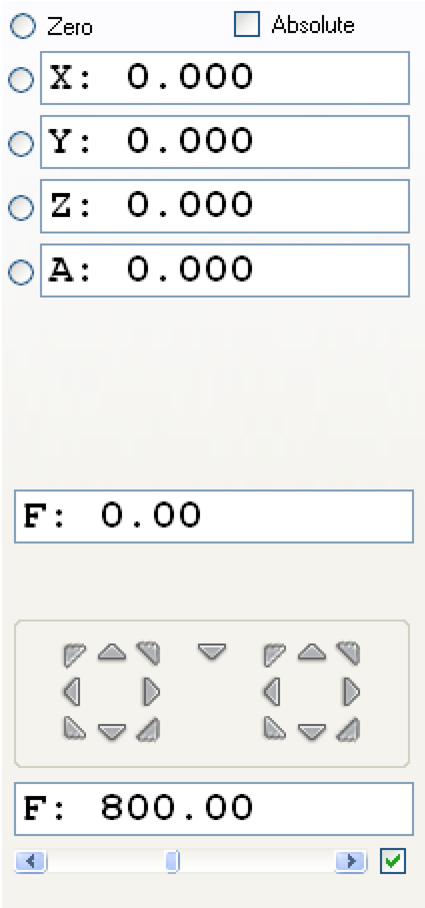
## 3.2 Main window




Components of main window are:

- A – position, state and program panels
- B – graphical program visualization display
- C – G-code panel
- D – manual data input (MDI) panel
- E – menus and toolbars
- F – status bar

### 3.2.1 Position panel




Position panel main features are:

- Current machine position coordinates
- Zero buttons
- Current machine speed
- Jog buttons
- Speed override control

Current machine position coordinates are displayed based on selected coordinate system. To display machine absolute coordinates check *Absolute* option.

Zero buttons are used to set position to zero for one of for all axes.

Speed is displayed in 'default units per minute'. Default units are set in settings.

Jog buttons are used to move machine to desired position. Jogging speed is set with speed override slider.

Speed override value is set with slider. Currrent value is displayed above slider. Speed override is used only when it is enabled with check box next to slider.

### 3.2.2 State panel

State panel shows information about:

- Current machine state
- Program simulation state

Displayed informations are:

- Units – millimeters or inches
- Plane – XY, YZ or ZX
- Mode – absolute or relative
- Feed, Traverse - speed
- Flood, Mist, Spindle - output
- Axes – coordinates
- Offset – current origin and coordinates
- Tool – tool number and offsets
- Line – G-code line number

| Current | |
|---|---|
| Units | Millimeters |
| Plane | XY |
| Mode | Absolute |
| Feed | 800.00 ! |
| Traverse | 1000.00 ! |
| Feed (O) | 800.00 |
| Traverse (O) | 800.00 |
| Flood | Unknown |
| Mist | Unknown |
| Spindle | Stopped |
| **Axes** | |
| X | 0.000 |
| Y | 0.000 |
| Z | 0.000 |
| A | 0.000 |
| **Offset** | |
| Current | 1 |
| Offset X | 0.000 |
| Offset Y | 0.000 |
| Offset Z | 0.000 |
| Offset A | 0.000 |

### 3.2.3 Program panel

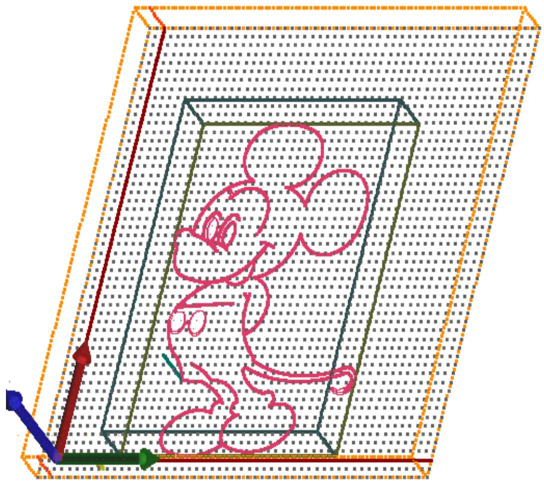
Program panel shows information about loaded program.

- Name – name of program
- Units – units used in this panel
- Min, Max – minimum and maximum values used in program (program extents)
- Min Feed, Max Feed – minimum and maximum values used for feed moves in program (cutting extents)
- Length – length of toolpath
- Time – estimated time needed to execute program
- Time (O) – estimated time needed to execute program if using speed override

| | |
|---|---|
| Name | mickey |
| Units | Millimeters |
| Min X | 16.819 |
| Max X | 74.310 |
| Min Y | 0.877 |
| Max Y | 77.692 |
| Min Z | 0.000 |
| Max Z | 25.000 |
| Min A | 0.000 |
| Max A | 0.000 |
| Min Feed X | 16.819 |
| Max Feed X | 74.310 |
| Min Feed Y | 0.877 |
| Max Feed Y | 77.692 |
| Min Feed Z | 0.000 |
| Max Feed Z | 0.000 |
| Min Feed A | 0.000 |
| Max Feed A | 0.000 |
| Length | 820.553 |
| Time | 00:03:04 |
| Time (O) | 00:01:01 |

### 3.2.4 Graphical program visualization display

This display shows 3D representation of machine, tool, toolpath,...



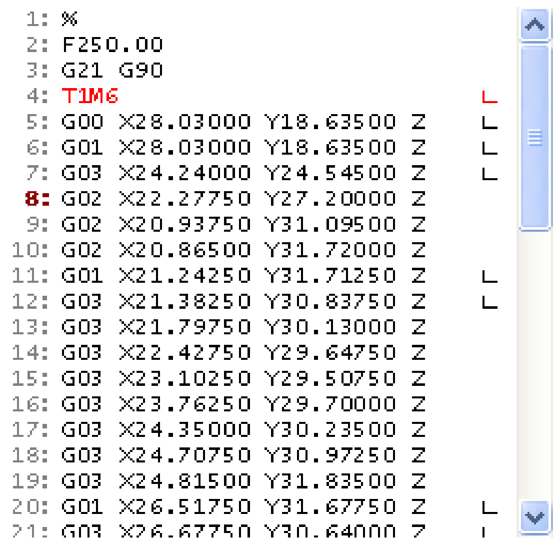
- Orange box – machine limits
- Gray grid – machine table
- Axis arrows – displayed coordinate system origin
- Red axis lines – selected coordinate system origin
- White line – feed toolpath
- Cyan line – traverse toolpath
- Red line – selected toolpath section
- Yellow cone – current machine position
- Orange cone – simulated position
- Dark green/gray cone – G28 and G30 positions
- Dark green/gray – program extents and program cutting extents

### 3.2.5 G-code panel

G-code panel shows current program lines. Lines are numbered.

Red lines represent pause. Machine willl pause execution at these lines,

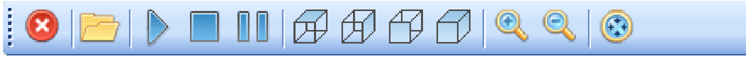
Lines with dark red line number are bookmarked.




### 3.2.6 Manual data input (MDI) panel

MDI is text input box, which allows manual G-code input.
For example typing *G0 X100* will move machine at traverse rate to X100 position.

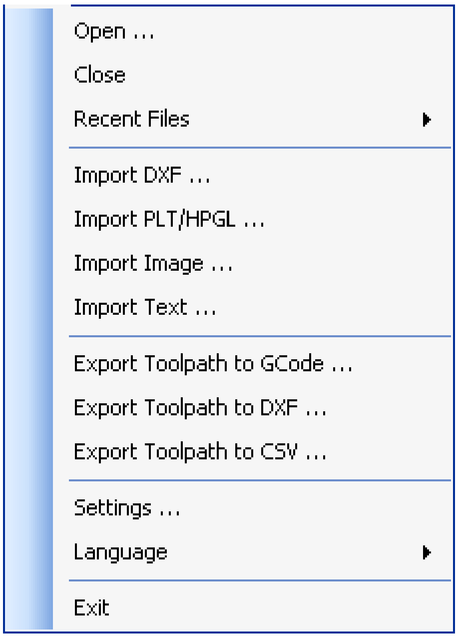
### 3.2.7 Toolbar



Toolbar has buttons for quick access to:

- Emergency stop
- Open program
- Start executing program
- Stop program execution
- Pause program execution
- Top View - display program as viewed from top
- Side View - display program as viewed from side
- Front View - display program as viewed from front
- Perspective View - display program in perspective
- Zoom In - zoom in display to view details
- Zoom Out - zoom out display to view larger part
- Center View - center display at program origin

### 3.2.8 File menu




#### 3.2.8.1 Open

Open new G-code, DXF or image file. Application will create toolpath and machine program from this file.

#### 3.2.8.2 Close

Close currently open program.

#### 3.2.8.3 Recent Files

Display a list of recently opened files.

#### 3.2.8.4 Import DXF

Open new DXF file. After file is selected, following parameters should be specified:

- Height – safe height of tool to move from one element to another
- Scale – scale of opened image according to original size,
- Sort – should be toolpath optimized (to avoid unnecessary moves of tool).

Application will create toolpath and machine program from this file.

#### 3.2.8.5 Import Image

Open new image file. After image is selected, following parameters should be specified:

- Size – size of piece
- Height – safe height of tool
- Invert – positive or negative relief
- Diameter – diameter of used tool
- Detail – interpolation detail level
- Method – parameter, on which depends the height (R – Red, G – Green, B – Blue, H – Hue, S – Saturation, L – Lightness)

Application will create toolpath and machine program from this file according to selected method.

#### 3.2.8.6 Import Text

After text is submitted, following parameters should be specified:

- Height – safe height of tool to move from one element to another
- Scale – scale of opened image according to original size,
- Sort – should be toolpath optimized (to avoid unnecessary moves of tool).

Application will create toolpath and machine program from written text.

#### 3.2.8.7 Export Toolpath To G-code

Export toolpath to G-code file. Extension of file should be written manually.

#### 3.2.8.8 Export Toolpath to DXF

Export toolpath to DXF file. Created sketch contains all tool paths, including moves of tool from one element to another on safe height.

#### 3.2.8.9 Export Toolpath to CSV

Export toolpath to CSV file. Created list of points contains points, that are visited during program execution.

#### 3.2.8.10 Settings

Display settings dialog, where general settings, axes, used tools, tool changes, used materials and coordinate systems are specified (see 3.3).

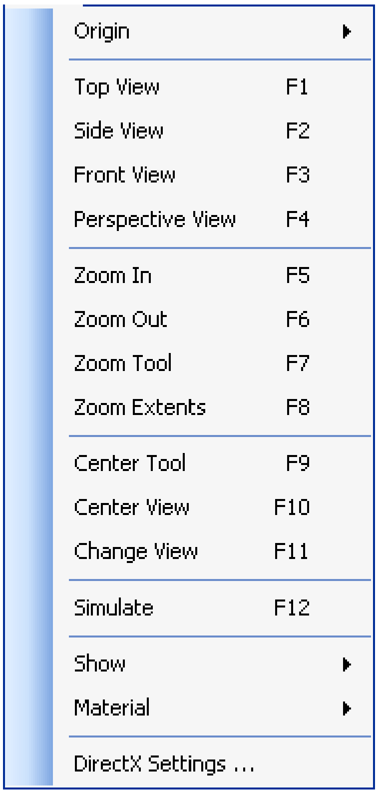
#### 3.2.8.11 Language

Select language of application from list of supported languages.

#### 3.2.8.12 Exit

Exit application.

## 3.2.9 View menu




### 3.2.9.1 Origin

Show defined origins on display.

### 3.2.9.2 Top View

Display program as viewed from top.

### 3.2.9.3 Side View

Display program as viewed from side.

### 3.2.9.4 Front View

Display program as viewed from front.

### 3.2.9.5 Perspective View

Display program in perspective.

### 3.2.9.6 Zoom In

Zoom in display to view details.

### 3.2.9.7 Zoom Out

Zoom out display to view larger part.

### 3.2.9.8 Zoom Tool

Center and zoom display to view detail at current tool position.

### 3.2.9.9 Zoom Extents

Center and zoom display to view whole program.

### 3.2.9.10 Center Tool

Center display at current tool position.

### 3.2.9.11 Center View

Center display at program origin.

### 3.2.9.12 Change View

Switch between horizontal and vertical position of G-code panel.

### 3.2.9.13 Simulate

Animate simulation tool handle on display.

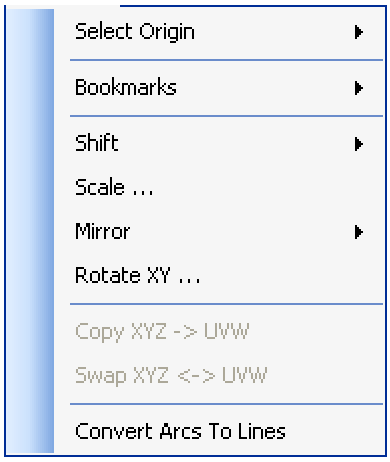
#### 3.2.9.14 Show

- Grid – show/hide grid
- Axes – show/hide axes
- Working Area – show/hide machine working area
- Tool – show/hide tool
- Tool Handle – show/hide tool handle. Only small tool is visible on display when tool handle is not shown..
- Tool G28, G30 – show/hide G28 and G30 positions.
- Extents – show/hide box representing program extents.
- Projection – toggles between orthogonal and perspective projection

#### 3.2.9.15 Material

Show defined materials to select one of them. Properties of selected material are used for tool length measurement.
Materials are specified in Settings (see 3.3.6).
You can show material in wireframe or solid mode. In solid mode material texture is based on material type.

## 3.2.10 Program menu




### 3.2.10.1 Select Origin

Select origin and set it as current origin.

### 3.2.10.2 Bookmarks

Select bookmarked G-code lines and set/clear bookmark. Use of bookmarks is specified in section 3.2.11.5.

### 3.2.10.3 Shift

Shift G-code program. Examples:

- X: 10,0000 – 10 will be added to coordinate X (program moves to right)
- X: -10,0000 – 10 will be subtracted from coordinate X (program moves to left)

One of following options can be selected:

- User – for each used coordinate can be inserted value to shift program
- Extends to Zero – program will be moved to position zero
- Extends to Position – program will be moved to current tool position

### 3.2.10.4 Scale

Scale G-code program. For each axis in use insert a factor for scale a program. Examples:

- X: 4,000 – coordinate X is multiplied by factor 4

### 3.2.10.5 Mirror

Mirror G-code program according to selected axis. Program will remain at current position.

### 3.2.10.6 Rotate

Rotate G-code program in selected plain by entered degree value.

### 3.2.10.7 Swap XYZ ↔ UVW

Swap X, Y and Z coordinates with U, V and W. This is used in 4-axis hot wire machines. Example: Program is made to create left wing of plane. By use of swap command existing program creates right wing of plane.

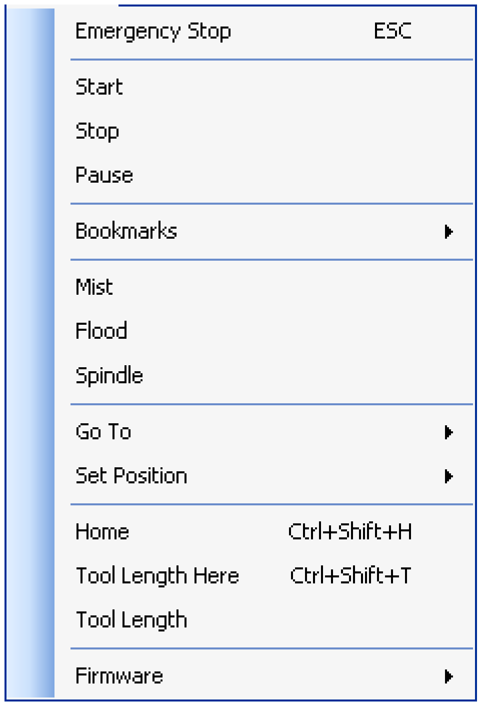
### 3.2.10.8 Convert Arcs To Lines

Convert arc moves to line moves

### 3.2.10.9 Convert To Lines

Convert program to line moves.

### 3.2.11 Machine menu




#### 3.2.11.1 Emergency Stop

Immediately send “STOP” commands to controller and stops executing program.

#### 3.2.11.2 Start

*Start* executing program.

#### 3.2.11.3 Stop

*Stop* executing program.

#### 3.2.11.4 Pause

Pause or resume execution of program.

#### 3.2.11.5 Bookmarks

Start executing program based on bookmarks:

- *Start - Only Bookmarked* – only bookmared lines will be executed
- *Start - Skip Bookmarked* – bookmarked lines will not be executed
- *Start - From Begining To Bookmark* – execution will end at first bookmark
- *Start - From Bookmark To End* – execution will start at first bookmark

#### 3.2.11.6 Mist

Mist output will be turned on or off.

#### 3.2.11.7 Flood

Flood output will be turned on or off.

#### 3.2.11.8 Spindle

Spindle output will be turned on or off.

#### 3.2.11.9 Go To Position

Move machine to specified position:

- *Zero* – Absolute machine zero
- *Park 1* – Park 1 position
- *Park 2* – Park 2 position
- *G28* – G28 position
- *G30* – G30 position
- *User …* – User defined position

#### 3.2.11.10 Set Position

Set machine current position to value:

- *Zero* – Absolute machine zero
- *Park 1* – Park 1 position
- *Park 2* – Park 2 position
- *G28* – G28 position
- *G30* – G30 position
- *User …* – User defined position

#### 3.2.11.11 Home

Execute homing procedure.

#### 3.2.11.12 Tool length here

Measures tool length at current XY position.

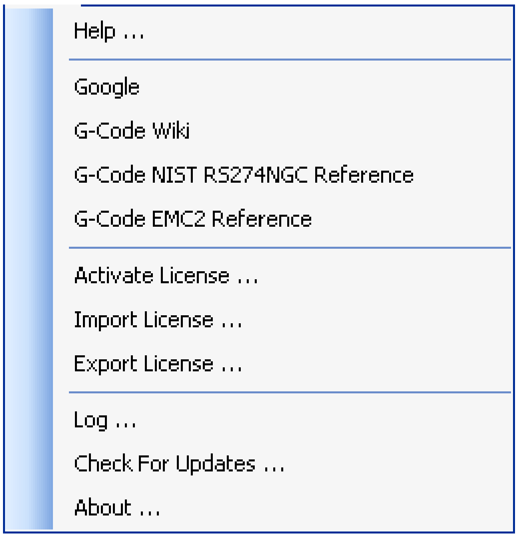
#### 3.2.11.13 Tool length

Execute tool length procedure.

#### 3.2.11.14 Firmware

Manually update or verify controller firmware.

### 3.2.12 Help menu




Help menu includes links to useful external web sites.

#### 3.2.12.1 Help

Open user manual.

#### 3.2.12.2 Activate License

Open program license activation window.

#### 3.2.12.3 Import License

Import license from file.

#### 3.2.12.4 Export License

Export license to file.

#### 3.2.12.5 Log

Open program log window. This is used when diagnosing problems.

#### 3.2.12.6 About

Display about window which includes information about software version, firmware version and hardware serial number if available.

### 3.2.13 Status bar

Status bar displays  controller buffer state, progress status and some other useful information.

## 3.3 Settings

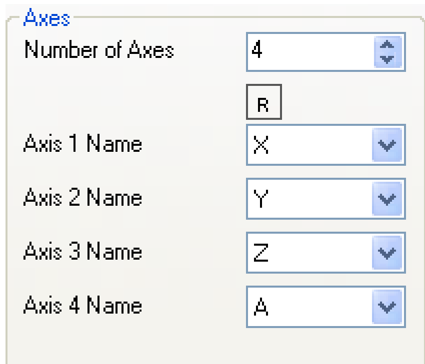
All machine settings and parameters are set in settings dialog.

### 3.3.1 General

#### 3.3.1.1 Units


Set machine default units. All settings will use this units.
On change of units all values will be recalculated.

#### 3.3.1.2 Axes



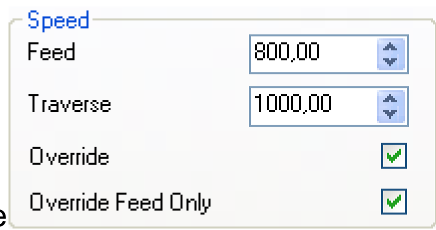

Set *Number of Axis* that you use. If your controller does not support so many axes they will be ignored.

Assign *Axis Name* to each axis. Most common names for 4-axis machine are X, Y, Z and A. Hot wire machines usually use X, Y, U and V.

Common used names of axes:

- X, Y, Z – normal axes
- A, B, C – rotary axes
- U, V, W – parallel axes (hot wire machine)

#### 3.3.1.3 Speed



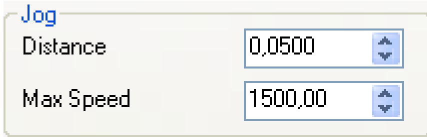

Set default *Feed* and *Traverse* speed in 'units per minute':

- Feed speed – speed of machine working in material
- Traverse speed – speed of machine moving on safe height to next position

This speed will be used unless program uses F word to specify speed or speed override is selected.

If *Override* is checked software will start with override enabled.
If *Override Feed Only* is checked only feed moves will be overridden and traverse moves will use default speed.

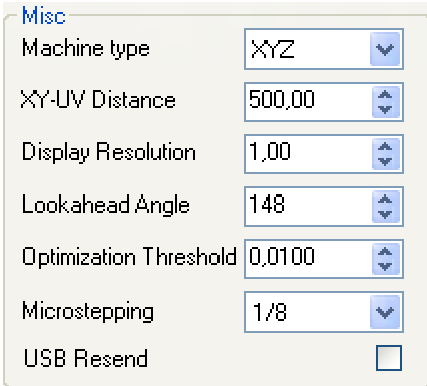
#### 3.3.1.4 Jog




Set jog *Distance* and maximum jog speed.

*Distance* is length that machine will travel on one single short click.
*Max Speed* sets value of speed when jog speed potentiometer is in maximal position.

#### 3.3.1.5 Misc




Set *Machine Type* for proper visualization on display:

- XYZ – classic mill or router style machine
- Hot wire – 4-axis foam cutting style machine
- Rotary – machine with rotary A axis

*XY-UV Distance* is distance between XY and UV plane on *Hot Wire* machine type.

*Display Resolution* sets detail level of display. Setting this to low value will degrade performance.

*Lookahead Angle* is used for calculating speed. If angle between two moves is larger than this value machine will move without lowering speed. If angle between two moves is less than this value machine will have to lower speed.
This is displayed on G-code panel with '└' and '┘' mark.

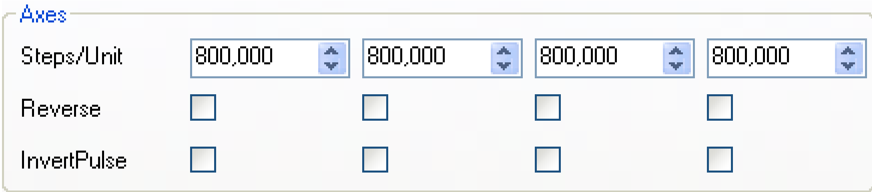
Optimization Threshold will set maximum detail level before optimization. Program will try to remove lines shorter than specified length. Setting this too low will produce jerky machine movement.

With *Microstepping* you tell controller, what microstepping your motor drivers use. Select closest available value.

*USB Resend* is used only for diagnostics.

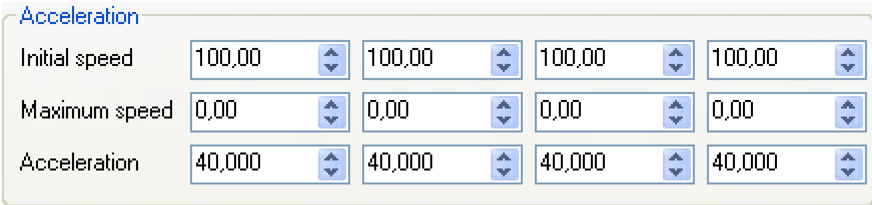
### 3.3.2 Axes 1

#### 3.3.2.1 Axes



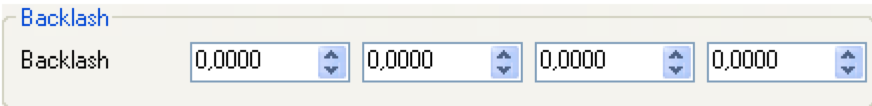

*Steps/Unit* is most important settings. Set its value tu number of steps that your machine needs for one unit of travel. This depends of motor, motor driver and screw.
*Reverse* will reverse motor direction.
*Invert Pulse* is used with some motor drivers which require 'negative' pulse.

#### 3.3.2.2 Acceleration



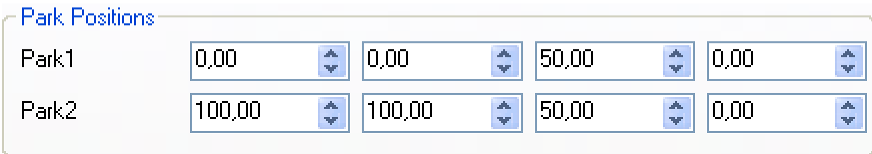

Set acceleration profile fom moves. Move will begin with *Initial speed* (units per minute) and will accelerate to target speed with *Acceleration* (units per second^2). *Maximum speed* that is allowed can also be set.

#### 3.3.2.3 Backlash



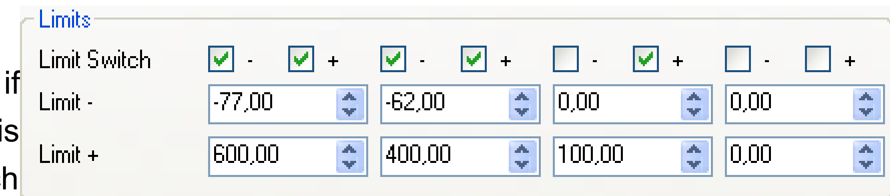

### 3.3.3 Axes 2

#### 3.3.3.1 Park Positions



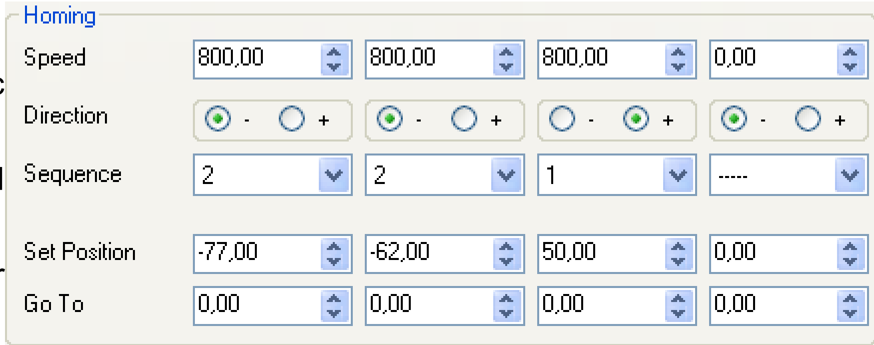

Set Park1 and Park2 positions. These can be used for many different things such as manual homing, tool change,...

#### 3.3.3.2 Limits




*Limit switch* checkbox sets if positive or negative limit switch is used on machine. If limit switch is used and it is hit during machine movement machine will stop. Exception are hardware jog buttons which move machine regardless on limit switch and special procedures which use switches for special purposes.
*Limit+* and *Limit-* are used to display machine limits on 3D display panel.

#### 3.3.3.3 Homing




These settings are for automatic homing procedure.

*Speed* is machine speed used for this procedure.

*Direction* sets if positive or negative direction is used.

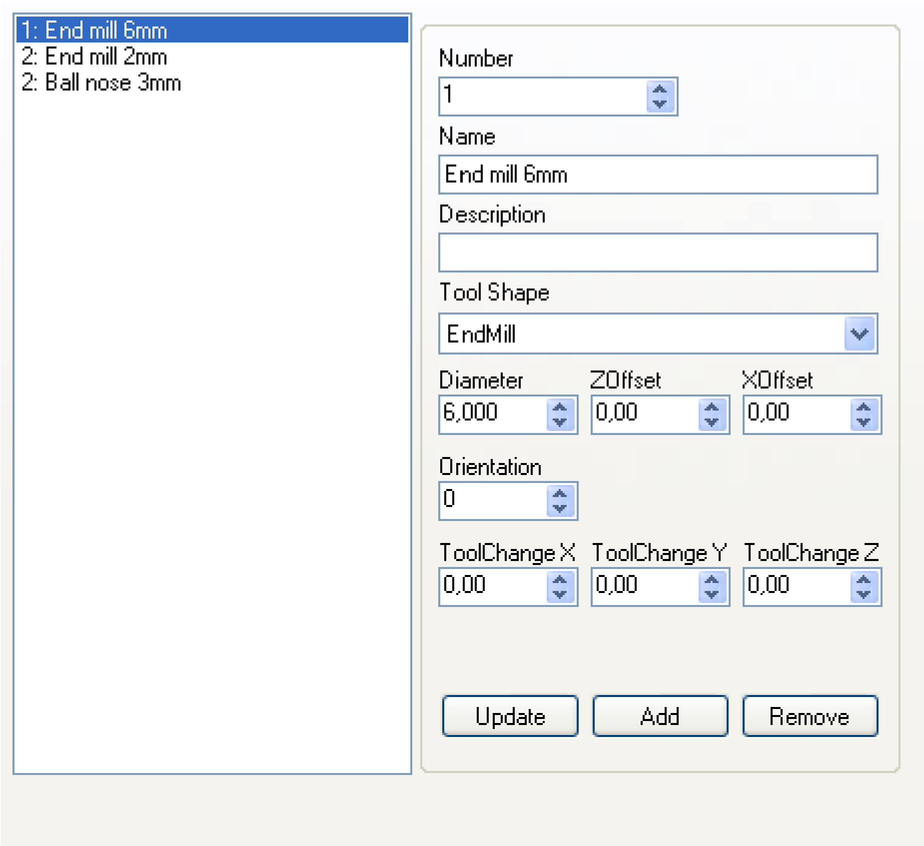
*Sequence* sets order in which axes are homed. '----' means that axis is not homed.

*Set Position* will be set when limit switch is hit.

*Go To* will move machine to this position afterwards.

Homing procedure will move specified axes to 'home' position. Set *Sequence* to specify axis order. Usually Z axis moves first, X and Y move second, other axes are not moved. Axis will move with set *Speed* and *Direction* until limit switch is hit.  When switch is hit *Set Position* will set coordinates of limit switch. After this, machine will *Go To* specified coordinates.

### 3.3.4 Tools



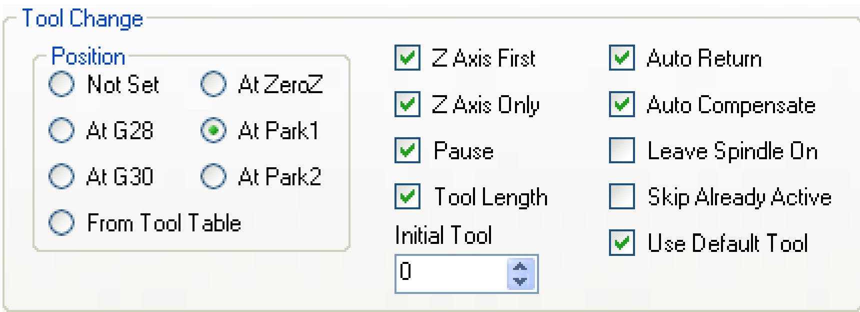

Tool table settings:

- *Number* – number of tool
- *Name* – name if tool
- *Description* – tool description
- *Tool Shape* – tool shape
- *Diameter* – tool diameter
- *Z Offset* – Z axis offset for tool length compensation
- *X Offset* – X axis offset for tool length compensation
- *Orientation* – tool orientation
- *Tool Change* – position for tool change procedure

*Update* button updates selected tool with new data.
*Add* button adds new tool to tool table.
*Remove* button removes tool from tool table.

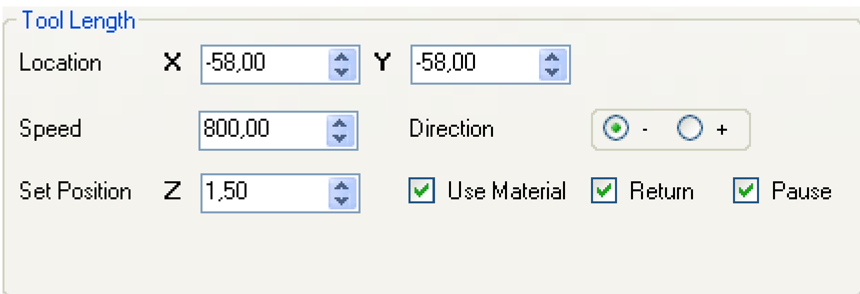
## 3.3.5 Tool Change

### 3.3.5.1 Tool Change



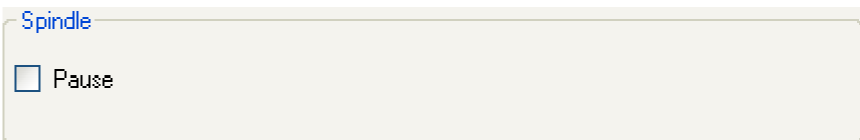

These settings determine what should machine do when toolchange command (M6) is executed.
*Position* sets where toolchange should occur.
*Z Axis First* will move Z axis before other axis so that movement is done at safe height.
*Z Axis Only* will move only Z axis. When you change tool manualy this is usualy enough.
*Pause* will pause machine at tool change. This will give you time to change tool.
*Tool length* will start tool length measurment procedure after tool change.
*Auto Return* will return machine to position before tool change after tool change is completed.
*Auto Compensate* will automaticaly adjust tool offset if tool length compensation (G43) is active
*Leave Spindle On* will not turn spindle off at tool change.
*Skip Already Active* will not perform tool change if tool is already active.
*Use Default Tool* will use tool with number zero if requested tool is not in tool table.
*Initial tool* is tool number that is already mounted and active on machine at begining.

### 3.3.5.2 Tool Length



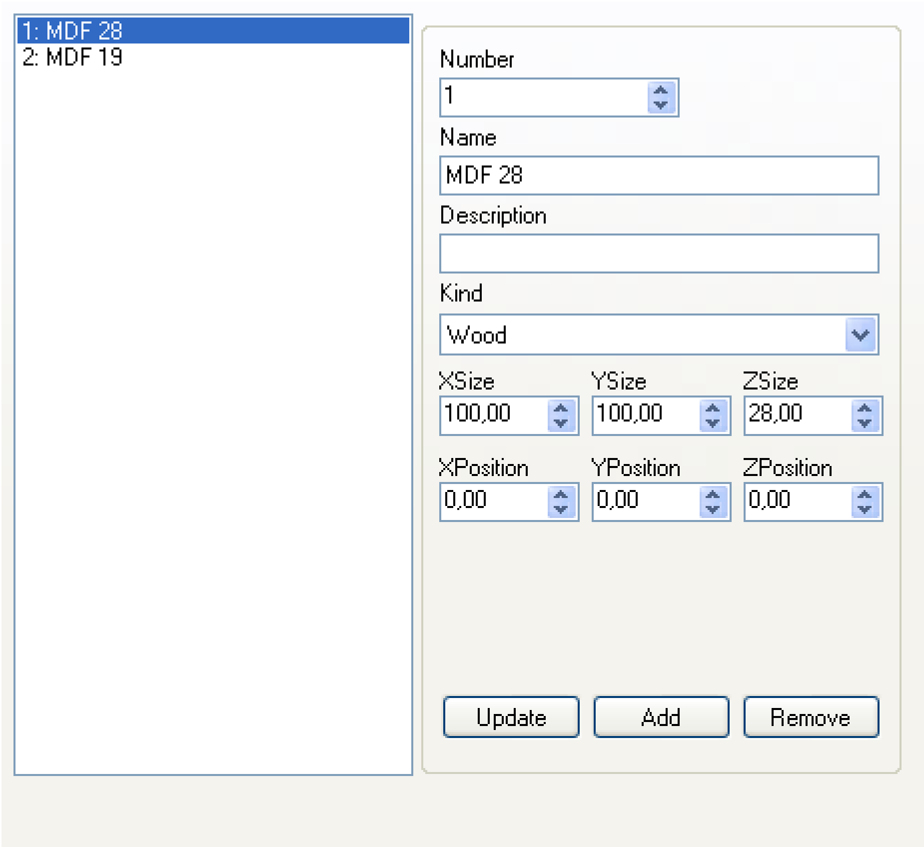

These settings determine how tool length procedure measures tool length.
*Location* is absolute coordinate of tool length sensor.
*Speed* is machine speed used for this procedure.
*Direction* sets if positive or negative Z direction is used.
*Set Position* is Z offset position of sensor (height of sensor).
*Use Material* will add offset from material table (material Z position + material Z size)
*Return* will move machine to position before tool length procedure.
*Pause* will pause machine at tool length.

### 3.3.5.3 Spindle




Set *Pause* if you want machine to pause on spindle commands.
This is can be used if you turn spindle on or off manually.

### 3.3.6 Materials

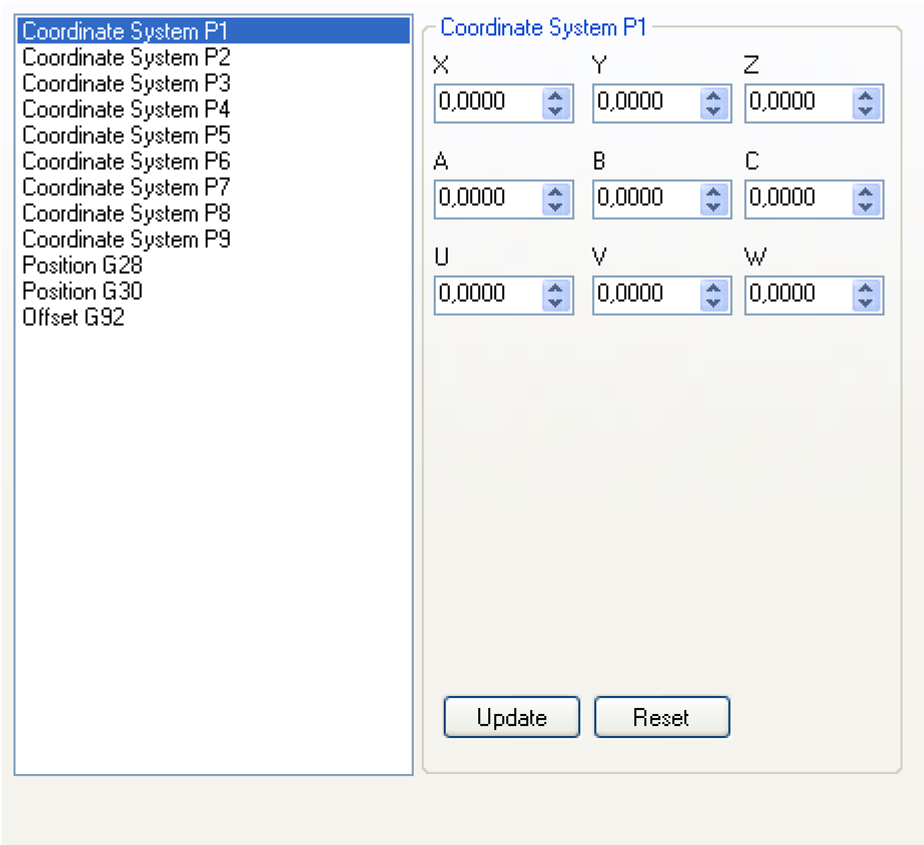



Material table is used for display and for offset in tool length procedure.

- *Number* – number of material
- *Name* – name if material
- *Description* – material description
- *Kind* – specify material kind
- *Size* – set material size
- *Position* – set material position in absolute coordinates

*Update* button updates selected material with new data.
*Add* button adds new material to material table.
*Remove* button removes material from material table.

### 3.3.7 Parameters




Parameter table is used for setting varous parameters such as coordinate system, G28 and G30 position, G92 offset. See G-code section for more information.

*Update* button updates selected parameter with new data.

*Reset* button resets parameter to default value.

## 3.4 FAQ

### 3.4.1 Firmware Update

Software will detect wrong firmware and will offer an option to update it.
You can also update firmware manually with *Machine/Firmware/Update* and verify it with *Machine/Firmware/Verify* commands.

### 3.4.2 Manual zeroing

Use commands *Machine/Go To Position* and *Machine/Set Position* to manually zero machine.

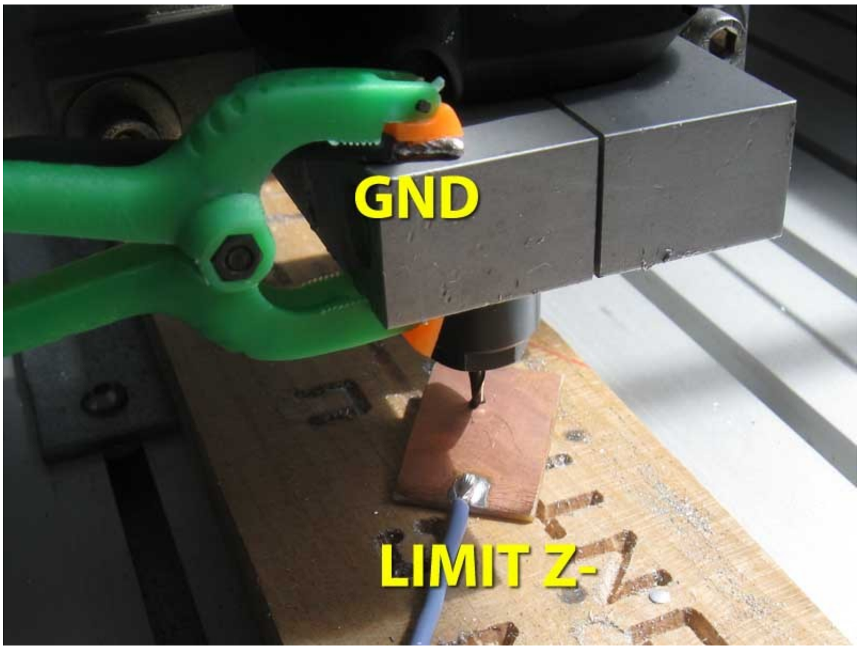
### 3.4.3 Homing procedure

Use commands *Machine/Home* to execute homing procedure.

### 3.4.4 Tool Length procedure

Use commands *Machine/Tool Length* or *Machine/Tool Length here* to start tool length measuring procedure.
In tool length procedure machine will move Z axis down (or up) until Limit Z switch is triggered.
Here is an example how to connect tool length switch made from piece of PCB.

- connect GND pin of *Limit* connector to toolbit. Usualy it is enough to connect it do spindle holder
- connect Limit Z- pin (LIMIT 5) to a piece of copper PCB



# 4 G-Code

G-code interpreter used by CNC USB controller is implemented according to National Institute of Standards and Technology RS274NGC Interpreter - Version 3 specification.
Full documentation of RS274NGC is available at web page:
http://www.isd.mel.nist.gov/personnel/kramer/pubs/RS274NGC_3.web/RS274NGC_3TOC.html

## 4.1 Overview

The RS274/NGC language is based on lines of code. Each line (also called a "block") may include commands to a machining center to do several different things. Lines of code may be collected in a file to make a program.

A typical line of code consists of an optional line number at the beginning followed by one or more "words." A word consists of a letter followed by a number (or something that evaluates to a number). A word may either give a command or provide an argument to a command. For example, "G1 X3" is a valid line of code with two words. "G1" is a command meaning "move in a straight line at the programmed feed rate," and "X3" provides an argument value (the value of X should be 3 at the end of the move). Most RS274/NGC commands start with either G or M (for miscellaneous). The words for these commands are called "G codes" and "M codes."

The RS274/NGC language has no indicator for the start of a program. The Interpreter, however, deals with files. A single program may be in a single file, or a program may be spread across several files. A file may demarcated with percents in the following way. The first non-blank line of a file may contain nothing but a percent sign, "%", possibly surrounded by white space, and later in the file (normally at the end of the file) there may be a similar line. Demarcating a file with percents is optional if the file has an M2 or M30 in it, but is required if not. An error will be signaled if a file has a percent line at the beginning but not at the end. The useful contents of a file demarcated by percents stop after the second percent line. Anything after that is ignored.

The RS274/NGC language has two commands (M2 or M30), either of which ends a program. A program may end before the end of a file. Lines of a file that occur after the end of a program are not to be executed.

## 4.2 Format of a Line

A permissible line of input RS274/NGC code consists of the following, in order, with the restriction that there is a maximum (currently 256) to the number of characters allowed on a line.

1. an optional block delete character, which is a slash "/" .
2. an optional line number.
3. any number of words, parameter settings, and comments.
4. an end of line marker (carriage return or line feed or both).

Any input not explicitly allowed is illegal and will cause the Interpreter to signal an error.

Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. This makes some strange-looking input legal. The line "g0x +0. 12 34y 7" is equivalent to "g0 x+0.1234 y7", for example.

Blank lines are allowed in the input. They are to be ignored.

Input is case insensitive, except in comments. Any letter outside a comment may be in upper or lower case without changing the meaning of a line.

## 4.3 Line Number

A line number is the letter N followed by an integer (with no sign) between 0 and 99999 written with no more than five digits (000009 is not OK, for example). Line numbers may be repeated or used out of order, although normal practice is to avoid such usage. Line numbers may also be skipped, and that is normal practice. A line number is not required to be used, but must be in the proper place if used.

## 4.4 Word

A word is a letter other than N followed by a real value.

Words may begin with any of the letters shown below. Several letters (I, J, K, L, P, R) may have different meanings in different contexts. Permitted letters are:
A, B, C, D, F, G, H, I, J, K, L, M, N, P. Q, R, S, T, X, Y, Z

## 4.5 Number

A real value is some collection of characters that can be processed to come up with a number. A real value may be an explicit number (such as 341 or -0.8807), a parameter value, an expression, or a unary operation value. Definitions of these follow immediately. Processing characters to come up with a number is called "evaluating". An explicit number evaluates to itself.

The following rules are used for (explicit) numbers. In these rules a digit is a single character between 0 and 9.

A number consists of an optional plus or minus sign, followed by zero to many digits, followed, possibly, by one decimal point, followed by zero to many digits - provided that there is at least one digit somewhere in the number.

There are two kinds of numbers: integers and decimals. An integer does not have a decimal point in it, a decimal does.

Numbers may have any number of digits, subject to the limitation on line length. Only about seventeen significant figures will be retained, however (enough for all known applications).

A non-zero number with no sign as the first character is assumed to be positive.

Notice that initial (before the decimal point and the first non-zero digit) and trailing (after the decimal point and the last non-zero digit) zeros are allowed but not required. A number written with initial or trailing zeros will have the same value when it is read as if the extra zeros were not there.

Numbers used for specific purposes in RS274/NGC are often restricted to some finite set of values or some to some range of values. In many uses, decimal numbers must be close to integers; this includes the values of indexes (for parameters and carousel slot numbers, for example), M codes, and G codes multiplied by ten. A decimal number which is supposed be close to an integer is considered close enough if it is within 0.0001 of an integer.

## 4.6 Parameters

A parameter value is the pound character # followed by a real value. The real value must evaluate to an integer between 1 and 5399. The integer is a parameter number, and the value of the parameter value is whatever number is stored in the numbered parameter.

The # character takes precedence over other operations, so that, for example, "#1+2" means the number found by adding 2 to the value of parameter 1, not the value found in parameter 3. Of course, #[1+2] does mean the value found in parameter 3. The # character may be repeated; for example ##2 means the value of the parameter whose index is the (integer) value of parameter 2.

A parameter setting is the following four items one after the other: (1) a pound character #, (2) a real value which evaluates to an integer between 1 and 5399, (3) an equal sign =, and (4) a real value. For example "#3 = 15" is a parameter setting meaning "set parameter 3 to 15."

A parameter setting does not take effect until after all parameter values on the same line have been found. For example, if parameter 3 has been previously set to 15 and the line "#3=6 G1 x#3" is interpreted, a straight move to a point where x equals 15 will occur and the value of parameter 3 will be 6.

## 4.7 Expressions and Binary Operations

An expression is a set of characters starting with a left bracket [ and ending with a balancing right bracket ]. In between the brackets are numbers, parameter values, mathematical operations, and other expressions. An expression may be evaluated to produce a number. The expressions on a line are evaluated when the line is read, before anything on the line is executed. An example of an expression is [ 1 + cos[0] - [#3 ** [4.0/2]]].

Binary operations appear only inside expressions. Nine binary operations are defined. There are four basic mathematical operations: addition (+), subtraction (-), multiplication (*), and division (/). There are three logical operations: non-exclusive or (OR), exclusive or (XOR), and logical and (AND). The eighth operation is the modulus operation (MOD). The ninth operation is the "power" operation (**) of raising the number on the left of the operation to the power on the right.

The binary operations are divided into three groups. The first group is: power. The second group is: multiplication, division, and modulus. The third group is: addition, subtraction, logical non-exclusive or, logical exclusive or, and logical and. If operations are strung together (for example in the expression [2.0 / 3 * 1.5 - 5.5 / 11.0]), operations in the first group are to be performed before operations in the second group and operations in the second group before operations in the third group. If an expression contains more than one operation from the same group (such as the first / and * in the example), the operation on the left is performed first. Thus, the example is equivalent to: [((2.0 / 3) * 1.5) - (5.5 / 11.0)], which simplifies to [1.0 - 0.5], which is 0.5.

The logical operations and modulus are to be performed on any real numbers, not just on integers. The number zero is equivalent to logical false, and any non-zero number is equivalent to logical true.

## 4.8 Unary Operation Value

A unary operation value is either "ATAN" followed by one expression divided by another expression (for example "ATAN[2]/[1+3]") or any other unary operation name followed by an expression (for example "SIN[90]"). The unary operations are: ABS (absolute value), ACOS (arc cosine), ASIN (arc sine), ATAN (arc tangent), COS (cosine), EXP (e raised to the given power), FIX (round down), FUP (round up), LN (natural logarithm), ROUND (round to the nearest whole number), SIN (sine), SQRT (square root), and TAN (tangent). Arguments to unary operations which take angle measures (COS, SIN, and TAN) are in degrees. Values returned by unary operations which return angle measures (ACOS, ASIN, and ATAN) are also in degrees.

The FIX operation rounds towards the left (less positive or more negative) on a number line, so that FIX[2.8] =2 and FIX[-2.8] = -3, for example. The FUP operation rounds towards the right (more positive or less negative) on a number line; FUP[2.8] = 3 and FUP[-2.8] = -2, for example.

## 4.9 Comments and Messages

Printable characters and white space inside parentheses is a comment. A left parenthesis always starts a comment. The comment ends at the first right parenthesis found thereafter. Once a left parenthesis is placed on a line, a matching right parenthesis must appear before the end of the line. Comments may not be nested; it is an error if a left parenthesis is found after the start of a comment and before the end of the comment. Here is an example of a line containing a comment: "G80 M5 (stop motion)". Comments do not cause a machining center to do anything.

A comment contains a message if "MSG," appears after the left parenthesis and before any other printing characters. Variants of "MSG," which include white space and lower case characters are allowed. The rest of the characters before the right parenthesis are considered to be a message. Messages should be displayed on the message display device. Comments not containing messages need not be displayed there.

## 4.10 Item Repeats

A line may have any number of G words, but two G words from the same modal group may not appear on the same line.

A line may have zero to four M words. Two M words from the same modal group may not appear on the same line.

For all other legal letters, a line may have only one word beginning with that letter.

If a parameter setting of the same parameter is repeated on a line, "#3=15 #3=6", for example, only the last setting will take effect. It is silly, but not illegal, to set the same parameter twice on the same line.

If more than one comment appears on a line, only the last one will be used; each of the other comments will be read and its format will be checked, but it will be ignored thereafter. It is expected that putting more than one comment on a line will be very rare.

## 4.11 Item Order

The three types of item whose order may vary on a line (as given at the beginning of this section) are word, parameter setting, and comment. Imagine that these three types of item are divided into three groups by type.

The first group (the words) may be reordered in any way without changing the meaning of the line.

If the second group (the parameter settings) is reordered, there will be no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter will take effect. For example, after the line "#3=15 #3=6" has been interpreted, the value of parameter 3 will be 6. If the order is reversed to "#3=6 #3=15" and the line is interpreted, the value of parameter 3 will be 15.

If the third group (the comments) contains more than one comment and is reordered, only the last comment will be used.

If each group is kept in order or reordered without changing the meaning of the line, then the three groups may be interleaved in any way without changing the meaning of the line. For example, the line "g40 g1 #3=15 (foo) #4=-7.0" has five items and means exactly the same thing in any of the 120 possible orders (such as "#4=-7.0 g1 #3=15 g40 (foo)") for the five items.

## 4.12 Modal and Non-modal modes

In RS274/NGC, many commands cause a machining center to change from one mode to another, and the mode stays active until some other command changes it implicitly or explicitly. Such commands are called "modal". For example, if coolant is turned on, it stays on until it is explicitly turned off. The G codes for motion are also modal. If a G1 (straight move) command is given on one line, for example, it will be executed again on the next line if one or more axis words is available on the line, unless an explicit command is given on that next line using the axis words or canceling motion.

"Non-modal" codes have effect only on the lines on which they occur. For example, G4 (dwell) is non-modal.

## 4.13 Modal Groups

Modal commands are arranged in sets called "modal groups", and only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time - like measure in inches vs. measure in millimeters. A machining center may be in many modes at the same time, with one mode from each modal group being in effect.

The modal groups for G codes are:
group 1 = {G0, G1, G2, G3, G38.2, G80, G81, G82, G83, G84, G85, G86, G87, G88, G89} motion
group 2 = {G17, G18, G19} plane selection
group 3 = {G90, G91} distance mode
group 5 = {G93, G94} feed rate mode
group 6 = {G20, G21} units
group 7 = {G40, G41, G42} cutter radius compensation
group 8 = {G43, G49} tool length offset
group 10 = {G98, G99} return mode in canned cycles
group 12 = {G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3} coordinate system selection
group 13 = {G61, G61.1, G64} path control mode

The modal groups for M codes are:
group 4 = {M0, M1, M2, M30, M60} stopping
group 6 = {M6} tool change
group 7 = {M3, M4, M5} spindle turning
group 8 = {M7, M8, M9} coolant (special case: M7 and M8 may be active at the same time)
group 9 = {M48, M49} enable/disable feed and speed override switches

In addition to the above modal groups, there is a group for non-modal G codes:
group 0 = {G4, G10, G28, G30, G53, G92, G92.1, G92.2, G92.3}

For several modal groups, when a machining center is ready to accept commands, one member of the group must be in effect. There are default settings for these modal groups. When the machining center is turned on or otherwise re-initialized, the default values are automatically in effect.

Group 1, the first group on the table, is a group of G codes for motion. One of these is always in effect. That one is called the current motion mode.

It is an error to put a G-code from group 1 and a G-code from group 0 on the same line if both of them use axis words. If an axis word-using G-code from group 1 is implicitly in effect on a line (by having been activated on an earlier line), and a group 0 G-code that uses axis words appears on the line, the activity of the group 1 G-code is suspended for that line. The axis word-using G-codes from group 0 are G10, G28, G30, and G92.

## 4.14 G - Codes

G-codes of the RS274/NGC language are shown and described below.

### 4.14.1 G00 - Rapid Positioning (Linear Motion at Traverse Rate)

For rapid linear motion, program G0 X- Y- Z- A- B- C-, where all the axis words are optional, except that at least one must be used. The G0 is optional if the current motion mode is G0. This will produce coordinated linear motion to the destination point at the current traverse rate (or slower if the machine will not go that fast). It is expected that cutting will not take place when a G0 command is executing.
It is an error if:

- all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above.

### 4.14.2 G01 - Linear Interpolation (Linear Motion at Feed Rate)

For linear motion at feed rate (for cutting or not), program G1 X- Y- Z- A- B- C-, where all the axis words are optional, except that at least one must be used. The G1 is optional if the current motion mode is G1. This will produce coordinated linear motion to the destination point at the current feed rate (or slower if the machine will not go that fast).
It is an error if:

- all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above.

### 4.14.3 G02 - Circular/Helical Interpolation (CW)

### 4.14.4 G03 - Circular/Helical Interpolation (CCW) (Arc at Feed Rate)

A circular or helical arc is specified using either G2 (clockwise arc) or G3 (counterclockwise arc). The axis of the circle or helix must be parallel to the X, Y, or Z-axis of the machine coordinate system. The axis (or, equivalently, the plane perpendicular to the axis) is selected with G17 (Z-axis, XY-plane), G18 (Y-axis, XZ-plane), or G19 (X-axis, YZ-plane). If the arc is circular, it lies in a plane parallel to the selected plane.

Two formats are allowed for specifying an arc. We will call these the center format and the radius format. In both formats the G2 or G3 is optional if it is the current motion mode.

- **Radius Format Arc**

In the radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. Program G2 X- Y- Z- A- B- C- R- (or use G3 instead of G2). R is the radius. The axis words are all optional except that at least one of the two words for the axes in the selected plane must be used. The R number is the radius. A positive radius indicates that the arc turns through 180 degrees or less, while a negative radius indicates a turn of 180 degrees to 359.999 degrees. If the arc is helical, the value of the end point of the arc on the coordinate axis parallel to the axis of the helix is also specified.
It is an error if:

- both of the axis words for the axes of the selected plane are omitted,
- the end point of the arc is the same as the current point.

It is not good practice to program radius format arcs that are nearly full circles or are semicircles (or nearly semicircles) because a small change in the location of the end point will produce a much larger change in the location of the center of the circle (and, hence, the middle of the arc). The magnification effect is large enough that rounding error in a number can produce out-of-tolerance cuts. Nearly full circles are outrageously bad, semicircles (and nearly so) are only very bad. Other size arcs (in the range tiny to 165 degrees or 195 to 345 degrees) are OK.

Here is an example of a radius format command to mill an arc: G17 G2 x 10 y 15 r 20 z 5.

That means to make a clockwise (as viewed from the positive Z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=10, Y=15, and Z=5, with a radius of 20. If the starting value of Z is 5, this is an arc of a circle parallel to the XY-plane; otherwise it is a helical arc.

- **Center Format Arc**

In the center format, the coordinates of the end point of the arc in the selected plane are specified along with the offsets of the center of the arc from the current location. In this format, it is OK if the end point of the arc is the same as the current point. It is an error if:

· when the arc is projected on the selected plane, the distance from the current point to the center differs from the distance from the end point to the center by more than 0.0002 inch (if inches are being used) or 0.002 millimeter (if millimeters are being used).

When the XY-plane is selected, program G2 X- Y- Z- A- B- C- I- J- (or use G3 instead of G2). The axis words are all optional except that at least one of X and Y must be used. I and J are the offsets from the current location (in the X and Y directions, respectively) of the center of the circle. I and J are optional except that at least one of the two must be used. It is an error if:

- X and Y are both omitted,
- I and J are both omitted.

When the XZ-plane is selected, program G2 X- Y- Z- A- B- C- I- K- (or use G3 instead of G2). The axis words are all optional except that at least one of X and Z must be used. I and K are the offsets from the current location (in the X and Z directions, respectively) of the center of the circle. I and K are optional except that at least one of the two must be used. It is an error if:

- X and Z are both omitted,
- I and K are both omitted.

When the YZ-plane is selected, program G2 X- Y- Z- A- B- C- J- K- (or use G3 instead of G2). The axis words are all optional except that at least one of Y and Z must be used. J and K are the offsets from the current location (in the Y and Z directions, respectively) of the center of the circle. J and K are optional except that at least one of the two must be used.

It is an error if:

- Y and Z are both omitted,
- J and K are both omitted.

Here is an example of a center format command to mill an arc: G17 G2 X10 Y16 I3 J4 Z9.

That means to make a clockwise (as viewed from the positive z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=10, Y=16, and Z=9, with its center offset in the X direction by 3 units from the current X location and offset in the Y direction by 4 units from the current Y location. If the current location has X=7, Y=7 at the outset, the center will be at X=10, Y=11. If the starting value of Z is 9, this is a circular arc; otherwise it is a helical arc. The radius of this arc would be 5.

In the center format, the radius of the arc is not specified, but it may be found easily as the distance from the center of the circle to either the current point or the end point of the arc.

### 4.14.5 G04 - Dwell

For a dwell, program G4 P- . This will keep the axes unmoving for the period of time in seconds specified by the P number. It is an error if:

- the P number is negative.

### 4.14.6 G10 L1 – Set Tool Table

G10 L1 P[tool number] R[radius] X[offset] Z[offset] Q[orientation]
Program a G10 L1 to set a tool table entry from a program. G10 L1 reloads the tool table.
It is an error if:

- cutter compensation is on

### 4.14.7 G10 L2 - Coordinate System Origin Setting

You can set the offsets of the nine program coordinate systems using G10 L2 Pn (n is the number of the coordinate system) with values for the axes in terms of the absolute coordinate system.

To set the coordinate values for the origin of a coordinate system, program
G10 L2 P - X- Y- Z- A- B- C-, where the P number must evaluate to an integer in the range 1 to 9 (corresponding to G54 to G59.3) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.
It is an error if:

- the P number does not evaluate to an integer in the range 1 to 9.

If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.

Example: G10 L2 P1 x 3.5 y 17.2 sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Y is 17.2 (in absolute coordinates). The Z coordinate of the origin (and the coordinates for any rotational axes) are whatever those coordinates of the origin were before the line was executed.

### 4.14.8 G10 L9 – Set Machine Position Without Move

Examples:
G10 L2 P1 X- Y- Z- A- B- C-  Machine will set position to coordinates defined with axis words
G10 L2 P28 Machine will set position to coordinates defined as G28 home
G10 L2 P30 Machine will set position to coordinates defined as G30 home
G10 L2 P92 Machine will set position to coordinates defined as G92 offset

### 4.14.9 G17 - XY-Plane Selection

### 4.14.10 G18 - XZ-Plane Selection

### 4.14.11 G19 - YZ-Plane Selection

Program G17 to select the XY-plane.
Program G18 to select the XZ-plane.
Program G19 to select the YZ-plane.

### 4.14.12 G20 - Inch System Selection

### 4.14.13 G21 - Millimeter System Selection

Program G20 to use inches for length units.
Program G21 to use millimeters.

It is usually a good idea to program either G20 or G21 near the beginning of a program before any motion occurs, and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units.

### 4.14.14 G28 - Return to Home

### 4.14.15 G30 - Return to Secondary Home

Two home positions are defined (by parameters 5161-5166 for G28 and parameters 5181-5186 for G30). The parameter values are in terms of the absolute coordinate system, but are in unspecified length units.

To return to home position by way of the programmed position, program G28 X- Y- Z- A- B- C- (or use G30). All axis words are optional. The path is made by a traverse move from the current position to the programmed position, followed by a traverse move to the home position. If no axis words are programmed, the intermediate point is the current point, so only one move is made.

### 4.14.16 G38.2 - Straight Probe

(not implemented)
Program G38.2 X- Y- Z- A- B- C- to perform a straight probe operation. The rotational axis words are allowed, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move. The linear axis words are optional, except that at least one of them must be used. The tool in the spindle must be a probe.
It is an error if:

- the current point is less than 0.254 millimeter or 0.01 inch from the programmed point.
- G38.2 is used in inverse time feed rate mode,
- any rotational axis is commanded to move,
- no X, Y, or Z-axis word is used.

In response to this command, the machine moves the controlled point (which should be at the end of the probe tip) in a straight line at the current feed rate toward the programmed point. If the probe trips, the probe is retracted slightly from the trip point at the end of command execution. If the probe does not trip even after overshooting the programmed point slightly, an error is signaled.

### 4.14.17 G40 - Cancel Cutter Radius Compensation

To turn cutter radius compensation off, program G40. It is OK to turn compensation off when it is already off.

### 4.14.18 G41 - Start Cutter Radius Compensation Left

### 4.14.19 G42 - Start Cutter Radius Compensation Right

Cutter radius compensation may be performed only if the XY-plane is active.

To turn cutter radius compensation on left (i.e., the cutter stays to the left of the programmed path when the tool radius is positive), program G41 D- . To turn cutter radius compensation on right (i.e., the cutter stays to the right of the programmed path when the tool radius is positive), program G42 D- . The D word is optional; if there is no D word, the radius of the tool currently in the spindle will be used. If used, the D number should normally be the slot number of the tool in the spindle, although this is not required. It is OK for the D number to be zero; a radius value of zero will be used.

It is an error if:

- the D number is not an integer, is negative or is larger than the number of carousel slots,
- the XY-plane is not active,
- cutter radius compensation is commanded to turn on when it is already on.

### 4.14.20 G43 - Tool Length Offset

To use a tool length offset, program G43 H-, where the H number is the desired index in the tool table. It is expected that all entries in this table will be positive. The H number should be, but does not have to be, the same as the slot number of the tool currently in the spindle. It is OK for the H number to be zero; an offset value of zero will be used.
It is an error if:

- the H number is not an integer, is negative, or is larger than the number of carousel slots.

It is OK to program using the same offset already in use. It is also OK to program using no tool length offset if none is currently being used.

### 4.14.21 G49 - Cancel Tool Length Offset

To use no tool length offset, program G49.

### 4.14.22 G53 - Motion in Machine Absolute Coordinates

You can make straight moves in the absolute machine coordinate system by using G53 with either G0 or G1.

For linear motion to a point expressed in absolute coordinates, program G1 G53 X- Y- Z- A- B- C- (or use G0 instead of G1), where all the axis words are optional, except that at least one must be used. The G0 or G1 is optional if it is the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. This will produce coordinated linear motion to the programmed point. If G1 is active, the speed of motion is the

current feed rate (or slower if the machine will not go that fast). If G0 is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).
It is an error if:

- G53 is used without G0 or G1 being active,
- G53 is used while cutter radius compensation is on.

### 4.14.23 G54 - Use Preset Work Coordinate System 1

### 4.14.24 G55 - Use Preset Work Coordinate System 2

### 4.14.25 G56 - Use Preset Work Coordinate System 3

### 4.14.26 G57 - Use Preset Work Coordinate System 4

### 4.14.27 G58 - Use Preset Work Coordinate System 5

### 4.14.28 G59 - Use Preset Work Coordinate System 6

### 4.14.29 G59.1 - Use Preset Work Coordinate System 7

### 4.14.30 G59.2 - Use Preset Work Coordinate System 8

### 4.14.31 G59.3 - Use Preset Work Coordinate System 9

You can select one of the nine systems by using G54, G55, G56, G57, G58, G59, G59.1, G59.2, or G59.3. It is not possible to select the absolute coordinate system directly.

To select coordinate system 1, program G54, and similarly for other coordinate systems.
The system-number-G-code pairs are:
(1-G54), (2-G55), (3-G56), (4-G57), (5-G58), (6-G59), (7-G59.1), (8-G59.2), and (9-G59.3).
It is an error if:

- one of these G-codes is used while cutter radius compensation is on.

### 4.14.32 G61 - Set Path Control Mode: Exact Path

### 4.14.33 G61.1 - set Path Control Mode: Exact Stop

### 4.14.34 G64 - Set Path Control Mode: Continuous

(not implemented)
Program G61 to put the machining center into exact path mode, G61.1 for exact stop mode, or G64 for continuous mode. It is OK to program for the mode that is already active.

### 4.14.35 G80 - Cancel Motion Mode (Including any Canned Cycle)

Program G80 to ensure no axis motion will occur. It is an error if:

- axis words are programmed when G80 is active, unless a modal group 0 G code is programmed which uses axis words.

#### 4.14.36 Canned Cycles (G80 - G89)

The canned cycles G81 through G89 have been implemented as described in this section. Two examples are given with the description of G81 below.

All canned cycles are performed with respect to the currently selected plane. Any of the three planes (XY, YZ, ZX) may be selected. Throughout this section, most of the descriptions assume the XY-plane has been selected. The behavior is always analogous if the YZ or XZ-plane is selected.

Rotational axis words are allowed in canned cycles, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move.

All canned cycles use X, Y, R, and Z numbers in the NC code. These numbers are used to determine X, Y, R, and Z positions. The R (usually meaning retract) position is along the axis perpendicular to the currently selected plane (Z-axis for XY-plane, X-axis for YZ-plane, Y-axis for XZ-plane). Some canned cycles use additional arguments.

For canned cycles, we will call a number "sticky" if, when the same cycle is used on several lines of code in a row, the number must be used the first time, but is optional on the rest of the lines. Sticky numbers keep their value on the rest of the lines if they are not explicitly programmed to be different. The R number is always sticky.

In incremental distance mode: when the XY-plane is selected, X, Y, and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ or XZ-plane is selected, treatment of the axis words is analogous. In absolute distance mode, the X, Y, R, and Z numbers are absolute positions in the current coordinate system.

The L number is optional and represents the number of repeats. L=0 is not allowed. If the repeat feature is used, it is normally used in incremental distance mode, so that the same sequence of motions is repeated in several equally spaced places along a straight line. In absolute distance mode, L > 1 means "do the same cycle in the same place several times," Omitting the L word is equivalent to specifying L=1. The L number is not sticky.

When L>1 in incremental mode with the XY-plane selected, the X and Y positions are determined by adding the given X and Y numbers either to the current X and Y positions (on the first go-around) or to the X and Y positions at the end of the previous go-around (on the repetitions). The R and Z positions do not change during the repeats.

The height of the retract move at the end of each repeat (called "clear Z" in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and the retract mode is G98, OLD_Z), or otherwise to the R position.
It is an error if:

- X, Y, and Z words are all missing during a canned cycle,
- a P number is required and a negative P number is used,
- an L number is used that does not evaluate to a positive integer,
- rotational axis motion is used during a canned cycle,
- inverse time feed rate is active during a canned cycle,
- cutter radius compensation is active during a canned cycle.

When the XY plane is active, the Z number is sticky, and it is an error if:

- the Z number is missing and the same canned cycle was not already active,
- the R number is less than the Z number.

When the XZ plane is active, the Y number is sticky, and it is an error if:

- the Y number is missing and the same canned cycle was not already active,
- the R number is less than the Y number.

When the YZ plane is active, the X number is sticky, and it is an error if:

- the X number is missing and the same canned cycle was not already active,
- the R number is less than the X number.

**Preliminary and In-Between Motion**
At the very beginning of the execution of any of the canned cycles, with the XY-plane selected, if the current Z position is below the R position, the Z-axis is traversed to the R position. This happens only once, regardless of the value of L.

In addition, at the beginning of the first cycle and each repeat, the following one or two moves are made:

- 1. a straight traverse parallel to the XY-plane to the given XY position,
- 2. a straight traverse of the Z-axis only to the R position, if it is not already at the R position.

If the XZ or YZ plane is active, the preliminary and in-between motions are analogous.

### 4.14.37 G81 - Canned Cycle: Drilling

The G81 cycle is intended for drilling. Program G81 X- Y- Z- A- B- C- R- L-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Retract the Z-axis at traverse rate to clear Z.

Example 1. Suppose the current position is (1, 2, 3) and the XY-plane has been selected, and the following line of NC code is interpreted.

G90 G81 G98 X4 Y5 Z1.5 R2.8

This calls for absolute distance mode (G90) and OLD_Z retract mode (G98) and calls for the G81 drilling cycle to be performed once. The X number and X position are 4. The Y number and Y position are 5. The Z number and Z position are 1.5. The R number and clear Z are 2.8. Old Z is 3. The following moves take place.

1. a traverse parallel to the XY-plane to (4,5,3)
2. a traverse parallel to the Z-axis to (4,5,2.8)
3. a feed parallel to the Z-axis to (4,5,1.5)
4. a traverse parallel to the Z-axis to (4,5,3)

Example 2. Suppose the current position is (1, 2, 3) and the XY-plane has been selected, and the following line of NC code is interpreted.

G91 G81 G98 X4 Y5 Z-0.6 R1.8 L3

This calls for incremental distance mode (G91) and OLD_Z retract mode (G98) and calls for the G81 drilling cycle to be repeated three times. The X number is 4, the Y number is 5, the Z number is -0.6 and the R number is 1.8. The initial X position is 5 (=1+4), the initial Y position is 7 (=2+5), the clear Z position is 4.8 (=1.8+3), and the Z position is 4.2 (=4.8-0.6). Old Z is 3.

The first move is a traverse along the Z-axis to (1,2,4.8), since old Z < clear Z.

The first repeat consists of 3 moves.

1. a traverse parallel to the XY-plane to (5,7,4.8)
2. a feed parallel to the Z-axis to (5,7, 4.2)
3. a traverse parallel to the Z-axis to (5,7,4.8)

The second repeat consists of 3 moves. The X position is reset to 9 (=5+4) and the Y position to 12 (=7+5).

1. a traverse parallel to the XY-plane to (9,12,4.8)
2. a feed parallel to the Z-axis to (9,12, 4.2)
3. a traverse parallel to the Z-axis to (9,12,4.8)

The third repeat consists of 3 moves. The X position is reset to 13 (=9+4) and the Y position to 17 (=12+5).

1. a traverse parallel to the XY-plane to (13,17,4.8)
2. a feed parallel to the Z-axis to (13,17, 4.2)

3. a traverse parallel to the Z-axis to (13,17,4.8)

### 4.14.38 G82 - Canned Cycle: Drilling with Dwell

The G82 cycle is intended for drilling. Program G82 X- Y- Z- A- B- C- R- L- P-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Retract the Z-axis at traverse rate to clear Z.

### 4.14.39 G83 - Canned Cycle: Peck Drilling

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a "delta" increment along the Z-axis. Program G83 X- Y- Z- A- B- C- R- L- Q-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
3. Rapid back out to the clear_z.
4. Rapid back down to the current hole bottom, backed off a bit.
5. Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
6. Retract the Z-axis at traverse rate to clear Z.

It is an error if:

- the Q number is negative or zero.

### 4.14.40 G84 - Canned Cycle: Right Hand Tapping

The G84 cycle is intended for right-hand tapping with a tap tool.
Program G84 X- Y- Z- A- B- C- R- L-

1. Preliminary motion, as described above.
2. Start speed-feed synchronization.
3. Move the Z-axis only at the current feed rate to the Z position.
4. Stop the spindle.
5. Start the spindle counterclockwise.
6. Retract the Z-axis at the current feed rate to clear Z.
7. If speed-feed synch was not on before the cycle started, stop it.
8. Stop the spindle.
9. Start the spindle clockwise.

The spindle must be turning clockwise before this cycle is used.
It is an error if:

- the spindle is not turning clockwise before this cycle is executed.

With this cycle, the programmer must be sure to program the speed and feed in the correct proportion to match the pitch of threads being made. The relationship is that the spindle speed equals the feed rate times the pitch (in threads per length unit). For example, if the pitch is 2 threads per millimeter, the active length units are millimeters, and the feed rate has been set with the command F150, then the speed should be set with the command S300, since 150 x 2 = 300.

If the feed and speed override switches are enabled and not set at 100%, the one set at the lower setting will take effect. The speed and feed rates will still be synchronized.

### 4.14.41 G85 - Canned Cycle: Boring, No Dwell, Feed Out

The G85 cycle is intended for boring or reaming, but could be used for drilling or milling. Program G85 X- Y- Z- A- B- C- R- L-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Retract the Z-axis at the current feed rate to clear Z.

### 4.14.42 G86 - Canned Cycle: Boring, Spindle Stop, Rapid Out

The G86 cycle is intended for boring. This cycle uses a P number for the number of seconds to dwell. Program G86 X- Y- Z- A- B- C- R- L- P-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Stop the spindle turning.
5. Retract the Z-axis at traverse rate to clear Z.
6. Restart the spindle in the direction it was going.

The spindle must be turning before this cycle is used.
It is an error if:

- the spindle is not turning before this cycle is executed.

### 4.14.43 G87 - Canned Cycle: Back Boring

The G87 cycle is intended for back boring.
Program G87 X- Y- Z- A- B- C- R- L- I- J- K-

The situation is that you have a through hole and you want to counterbore the bottom of hole. To do this you put an L-shaped tool in the spindle with a cutting surface on the UPPER side of its base. You stick it carefully through the hole when it is not spinning and is oriented so it fits through the hole, then you move it so the stem of the L is on the axis of the hole, start the spindle, and feed the tool upward to make the counterbore. Then you stop the tool, get it out of the hole, and restart it.

This cycle uses I and J numbers to indicate the position for inserting and removing the tool. I and J will always be increments from the X position and the Y position, regardless of the distance mode setting. This cycle also uses a K number to specify the position along the Z-axis of the controlled point top of the counterbore. The K number is a Z-value in the current coordinate system in absolute distance mode, and an increment (from the Z position) in incremental distance mode.

1. Preliminary motion, as described above.
2. Move at traverse rate parallel to the XY-plane to the point indicated by I and J.
3. Stop the spindle in a specific orientation.
4. Move the Z-axis only at traverse rate downward to the Z position.
5. Move at traverse rate parallel to the XY-plane to the X,Y location.
6. Start the spindle in the direction it was going before.
7. Move the Z-axis only at the given feed rate upward to the position indicated by K.
8. Move the Z-axis only at the given feed rate back down to the Z position.
9. Stop the spindle in the same orientation as before.
10. Move at traverse rate parallel to the XY-plane to the point indicated by I and J.
11. Move the Z-axis only at traverse rate to the clear Z.
12. Move at traverse rate parallel to the XY-plane to the specified X,Y location.
13. Restart the spindle in the direction it was going before.

When programming this cycle, the I and J numbers must be chosen so that when the tool is stopped in an oriented position, it will fit through the hole. Because different cutters are made differently, it may take some analysis and/or experimentation to determine appropriate values for I and J.

### 4.14.44 G88 - Canned Cycle: Boring, Spindle Stop, Manual Out

The G88 cycle is intended for boring. This cycle uses a P word, where P specifies the number of seconds to dwell. Program G88 X- Y- Z- A- B- C- R- L- P-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Stop the spindle turning.
5. Stop the program so the operator can retract the spindle manually.
6. Restart the spindle in the direction it was going.

### 4.14.45 G89 - Canned Cycle: Boring, Dwell, Feed Out

The G89 cycle is intended for boring. This cycle uses a P number, where P specifies the number of seconds to dwell. program G89 X- Y- Z- A- B- C- R- L- P-

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Retract the Z-axis at the current feed rate to clear Z.

### 4.14.46 G90 - Absolute Distance Mode

Interpretation of RS274/NGC code can be in one of two distance modes: absolute or incremental.
To go into absolute distance mode, program G90. In absolute distance mode, axis numbers (X, Y, Z, A, B, C) usually represent positions in terms of the currently active coordinate system.

### 4.14.47 G91 - Incremental Distance Mode

Interpretation of RS274/NGC code can be in one of two distance modes: absolute or incremental.
To go into incremental distance mode, program G91. In incremental distance mode, axis numbers (X, Y, Z, A, B, C) usually represent increments from the current values of the numbers.
I and J numbers always represent increments, regardless of the distance mode setting. K numbers represent increments in all but one usage (see G87), where the meaning changes with distance mode.

### 4.14.48 G92 - Offset Coordinate Systems and Set Parameters

You can offset the current coordinate system using G92 or G92.3. This offset will then apply to all nine program coordinate systems. This offset may be canceled with G92.1 or G92.2.

To make the current point have the coordinates you want (without motion), program G92 X- Y- Z- A- B- C- , where the axis words contain the axis numbers you want. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

- all axis words are omitted.

When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5216 are set to the X, Y, Z, A, B, and C-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 x7 sets the X-axis offset to -3, sets parameter 5211 to -3, and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the nine coordinate systems (those designated by G54 - G59.3). Thus all nine coordinate systems are affected by G92.

Being in incremental distance mode has no effect on the action of G92.

Non-zero offsets may be already be in effect when the G92 is called. If this is the case, the new value of each offset is A+B, where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 x9 is then programmed, the new X-axis offset is -5, which is calculated by [[7-9] + -3].

### 4.14.49 G92.1 - Cancel Offset Coordinate Systems and Set Parameters to Zero

To reset axis offsets to zero, program G92.1 or G92.2. G92.1 sets parameters 5211 to 5216 to zero.

### 4.14.50 G92.2 - Cancel Offset Coordinate Systems But Do Not Reset Parameters

To reset axis offsets to zero, program G92.1 or G92.2. G92.2 leaves parameters 5211 to 5216 values alone.

### 4.14.51 G92.3 - Apply Parameters to Offset Coordinate Systems

To set the axis offset to the values saved in parameters 5211 to 5219, program G92.3.

### 4.14.52 G93 - Inverse Time Feed Rate Mode

Two feed rate modes are recognized: units per minute and inverse time. Program G93 to start the inverse time mode.

In inverse time feed rate mode, an F word means the move should be completed in [one divided by the F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute.

When the inverse time feed rate mode is active, an F word must appear on every line which has a G1, G2, or G3 motion, and an F word on a line that does not have G1, G2, or G3 is ignored. Being in inverse time feed rate mode does not affect G0 (rapid traverse) motions.
It is an error if:

- inverse time feed rate mode is active and a line with G1, G2, or G3 (explicitly or implicitly) does not have an F word.

### 4.14.53 G94 - Units Per Minute Feed Rate Mode

Two feed rate modes are recognized: units per minute and inverse time. Program G94 to start the units per minute mode.

In units per minute feed rate mode, an F word is interpreted to mean the controlled point should move at a certain number of inches per minute, millimeters per minute, or degrees per minute, depending upon what length units are being used and which axis or axes are moving.

### 4.14.54 G98 - Initial level Return in Canned Cycles

When the spindle retracts during canned cycles, there is a choice of how far it retracts. Program G98 to retract perpendicular to the selected plane to the position that axis was in just before the canned cycle started (unless that position is lower than the position indicated by the R word, in which case use the R word position).

### 4.14.55 G99 R-point Level Return in Canned Cycles

When the spindle retracts during canned cycles, there is a choice of how far it retracts. Program G99 to retract perpendicular to the selected plane to the position indicated by the R word.

## 4.15 M - Codes

### 4.15.1 M0 - Program Stop

To stop a running program temporarily (regardless of the setting of the optional stop switch), program M0.

### 4.15.2 M1 - Optional Program Stop

To stop a running program temporarily (but only if the optional stop switch is on), program M1.

### 4.15.3 M2 - Program End

To end a program, program M2.

### 4.15.4 M30 - Program End, Pallet Shuttle, and Reset

To exchange pallet shuttles and then end a program, program M30.

### 4.15.5 M60 - Pallet Shuttle and Program Stop

To exchange pallet shuttles and then stop a running program temporarily (regardless of the setting of the optional stop switch), program M60.

### 4.15.6 M3 - Turn Spindle Clockwise

To start the spindle turning clockwise at the currently programmed speed, program M3.

### 4.15.7 M4 - Turn Spindle Counterclockwise

To start the spindle turning counterclockwise at the currently programmed speed, program M4.

### 4.15.8 M5 - Stop Spindle Turning

To stop the spindle from turning, program M5.

### 4.15.9 M6 - Tool Change

To change a tool in the spindle from the tool currently in the spindle to the tool most recently selected (using a T word), program M6.

### 4.15.10 M7 - Mist Coolant On

To turn mist coolant on, program M7.

### 4.15.11 M8 - Flood Coolant On

To turn flood coolant on, program M8.

### 4.15.12 M9 - Mist and Flood Coolant Off

To turn all coolant off, program M9.
It is always OK to use any of these commands, regardless of what coolant is on or off.

### 4.15.13 M48 - Enable Speed and Feed Overrides

### 4.15.14 M49 - Disable Speed and Feed Overrides

(not implemented)
To enable the speed and feed override switches, program M48. To disable both switches, program M49. It is OK to enable or disable the switches when they are already enabled or disabled.

## 4.16 Other Codes

### 4.16.1 F - Set Feed Rate

To set the feed rate, program F. The rate at which the controlled point or the axes move is nominally a steady rate which may be set by the user. For motion involving one or more of the X, Y, and Z axes (with or without simultaneous rotational axis motion), the feed rate means length units per minute along the programmed XYZ path, as if the rotational axes were not moving.

### 4.16.2 S - Set Spindle Speed

To set the speed in revolutions per minute (rpm) of the spindle, program S- . The spindle will turn at that speed when it has been programmed to start turning. It is OK to program an S word whether the spindle is turning or not. If the speed override switch is enabled and not set at 100%, the speed will be different from what is programmed. It is OK to program S0; the spindle will not turn if that is done.
It is an error if:

- the S number is negative

### 4.16.3 T - Select Tool

To select a tool, program T-, where the T number is the carousel slot for the tool. The tool is not changed until an M6 is programmed. The T word may appear on the same line as the M6 or on a previous line. It is OK, but not normally useful, if T words appear on two or more lines with no tool change. The carousel may move a lot, but only the most recent T word will take effect at the next tool change. It is OK to program T0; no tool will be selected. This is useful if you want the spindle to be empty after a tool change.
It is an error if:

- a negative T number is used,
- a T number larger than the number of slots in the carousel is used.

## 4.17 Sample G-code programs

### 4.17.1 Square

```
%
M3
G01 X0 Y0
G01 Z-3
G01 Z3
G01 X-150 Y-150
G01 Z-3
G01 X150 Y-150
G01 X150 Y150
G01 X-150 Y150
G01 X-150 Y-150
G01 Z3
G01 X0 Y0
M5
%
```

### 4.17.2 Circle

```
%
M3
G01 X0 Y0
G01 Z-3
G01 Z3
G01 X0 Y-150
G01 Z-3
G02 X0 Y-150 I0 J150
G01 Z3
G01 X0 Y0
G01 Z-3
G01 Z3
M5
%
```

### 4.17.3 One side of a ball in cage

```
%
G90
G21
#1=50 (width of box)
#2=2 (tool radius)
#3=4 (height of box)
#4=5 (resolution in degrees)
#5=#4 (counter)
#6=[[#1/2]+#2] (actual radius of circle - cutter radius + radius)
G01 X0 Y0
o140 do
  #8 = [[SIN[#5]*#6]*SIN[45]] ( X and Y position)
  #9 = [0-[[1-COS[#5]]*#6]] (Z position)
  G1 X[#8] Y[#8]
  Z[#9]
  G18 G02 X[0-#8] Z[#9] I[0-#8] K[0-[#9+[#6]]]
  G19 G03 Y[0-#8] Z[#9] J[0-#8] K[0-[#9+[#6]]]
  G18 G03 X[#8] Z[#9] I[#8] K[0-[#9+[#6]]]
  G19 G02 Y[#8] Z[#9] J[#8] K[0-[#9+[#6]]]
  #5 = [#5+#4]
o140 while [#8 LT [#1/2-#2-#3]]
G1 Z1
X0 Y0
%
```