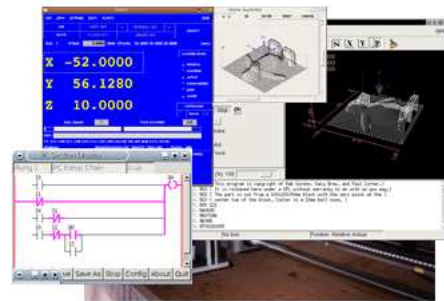


# The EMC2 (Enhanced Machine Control) User Handbook



## **EMC<sup>2</sup>**

**The Enhanced Machine Controller**



[www.linuxcnc.org](http://www.linuxcnc.org)

The EMC Team

4th May 2006

This handbook is a work in progress. If you are able to help with writing, editing, or graphic preparation please contact any member of the writing team or join and send an email to [emc-users@lists.sourceforge.net](mailto:emc-users@lists.sourceforge.net).

Copyright (c) 2000-6 LinuxCNC.org

---

Permission is granted to copy, distribute and/or modify this document under the terms of the GNU Free Documentation License, Version 1.1 or any later version published by the Free Software Foundation; with no Invariant Sections, no Front-Cover Texts, and one Back-Cover Text: "This EMC Handbook is the product of several authors writing for linuxCNC.org. As you find it to be of value in your work, we invite you to contribute to its revision and growth." A copy of the license is included in the section entitled "GNU Free Documentation License". If you do not find the license you may order a copy from Free Software Foundation, Inc. 59 Temple Place, Suite 330 Boston, MA 02111-1307

---

# Contents

<b>I</b>	<b>Introduction &amp; installing EMC2</b>	<b>8</b>
<b>1</b>	<b>The Enhanced Machine Control</b>	<b>9</b>
1.1	Introduction	9
1.2	The Big CNC Picture	9
1.3	Computer Operating Systems	10
1.4	History of the Software	10
1.5	How the EMC2 Works	11
1.5.1	Graphical User Interfaces	11
1.5.2	Motion Controller EMCMOT	12
1.5.3	Discrete I/O Controller EMCIO	13
1.5.4	Task Executor EMCTASK	14
1.6	Thinking Like a Machine Operator	15
1.6.1	Modes of Operation	16
1.6.2	Information Display	16
1.7	Thinking Like An Integrator	18
1.7.1	Units	18
1.7.2	Some things we may not want to change.	19
1.7.3	Some things we will need to change.	19
<b>2</b>	<b>Installing the EMC2 software</b>	<b>21</b>
2.1	Introduction	21
2.2	EMC Download Page	21
2.3	EMC2 install script - the easy way to install	22
2.4	Manual installing using apt commands.	22
<b>II</b>	<b>Configuring EMC2</b>	<b>23</b>
<b>3</b>	<b>Introduction</b>	<b>25</b>
3.1	What is HAL?	25
3.1.1	HAL is based on traditional system design techniques	25
3.1.1.1	Part Selection	25
3.1.1.2	Interconnection Design	25
3.1.1.3	Implementation	26
3.1.1.4	Testing	26
3.1.2	Summary	26
3.2	HAL Concepts	26

3.3 HAL components . . . . .	28
3.3.1 External programs with HAL hooks . . . . .	28
3.3.2 Internal Components . . . . .	28
3.3.3 Hardware drivers . . . . .	29
3.3.4 Utilities . . . . .	29
3.4 Tinkertoys, Erector Sets, Legos and the HAL . . . . .	29
3.4.1 Tower . . . . .	29
3.4.2 Erector Sets . . . . .	29
3.4.3 Tinkertoys . . . . .	30
3.4.4 A Lego Example . . . . .	30
3.5 Timing Issues In HAL . . . . .	31
3.6 Dynamic Linking and Configuration . . . . .	32
<b>4 HAL Tutorial</b>	<b>33</b>
4.1 Before we start . . . . .	33
4.1.1 Notation . . . . .	33
4.1.2 Root Privileges . . . . .	33
4.1.3 The RTAPI environment . . . . .	34
4.2 A Simple Example . . . . .	34
4.2.1 Loading a realtime component . . . . .	34
4.2.2 Examining the HAL . . . . .	35
4.2.3 Making realtime code run . . . . .	36
4.2.4 Changing parameters . . . . .	37
4.2.5 Saving the HAL configuration . . . . .	38
4.2.6 Restoring the HAL configuration . . . . .	38
4.3 Looking at the HAL with halmeter . . . . .	38
4.3.1 Starting halmeter . . . . .	39
4.3.2 Using halmeter . . . . .	39
4.4 A slightly more complex example. . . . .	42
4.4.1 Installing the components . . . . .	42
4.4.2 Connecting pins with signals . . . . .	43
4.4.3 Setting up realtime execution - threads and functions . . . . .	44
4.4.4 Setting parameters . . . . .	46
4.4.5 Run it! . . . . .	46
4.5 Taking a closer look with halscope. . . . .	46
4.5.1 Starting Halscope . . . . .	46
4.5.2 Hooking up the "scope probes" . . . . .	49
4.5.3 Capturing our first waveforms . . . . .	50
4.5.4 Vertical Adjustments . . . . .	51
4.5.5 Triggering . . . . .	51
4.5.6 Horizontal Adjustments . . . . .	53
4.5.7 More Channels . . . . .	54
<b>III Using EMC2</b>	<b>55</b>
<b>5 Using The TKEMC Graphical Interface</b>	<b>57</b>

<b>6</b>	<b>Using The MINI Graphical Interface</b>	<b>58</b>
6.1	Introduction	58
6.2	Screen layout	59
6.3	Menu Bar	59
6.4	Control Button Bar	61
6.4.1	MANUAL	61
6.4.2	AUTO	62
6.4.3	MDI	63
6.4.4	[FEEDHOLD] – [CONTINUE]	63
6.4.5	[ABORT]	63
6.4.6	[ESTOP]	63
6.5	Left Column	64
6.5.1	Axis Position Displays	64
6.5.2	Feedrate Override	64
6.5.3	Messages	65
6.6	Right Column	65
6.6.1	Program Editor	65
6.6.2	Backplot Display	65
6.6.3	Tool Page	66
6.6.4	Offset Page	67
6.7	Keyboard Bindings	67
6.7.1	Common Keys	67
6.7.2	Manual Mode	68
6.7.3	Auto Mode	68
6.8	Misc	68
<b>7</b>	<b>Machining Center Overview</b>	<b>69</b>
7.1	Mechanical Components	69
7.1.1	Linear Axes	69
7.1.2	Rotational axes	69
7.1.3	Spindle	70
7.1.4	Coolant	70
7.1.5	Pallet Shuttle	70
7.1.6	Tool Carousel	70
7.1.7	Tool Changer	70
7.1.8	Message Display	70
7.1.9	Feed and Speed Override Switches	70
7.1.10	Block Delete Switch	70
7.1.11	Optional Program Stop Switch	70
7.2	Control and Data Components	70
7.2.1	Linear Axes	70
7.2.2	Rotational Axes	71
7.2.3	Controlled Point	71
7.2.4	Coordinate Linear Motion	71
7.2.5	Feed Rate	71
7.2.6	Coolant	72

7.2.7 Dwell . . . . .	72
7.2.8 Units . . . . .	72
7.2.9 Current Position . . . . .	72
7.2.10 Selected Plane . . . . .	72
7.2.11 Tool Carousel . . . . .	72
7.2.12 Tool Change . . . . .	72
7.2.13 Pallet Shuttle . . . . .	72
7.2.14 Feed and Speed Override Switches . . . . .	73
7.2.15 Path Control Mode . . . . .	73
7.3 Interpreter Interaction with Switches . . . . .	73
7.3.1 Feed and Speed Override Switches . . . . .	73
7.3.2 Block Delete Switch . . . . .	73
7.3.3 Optional Program Stop Switch . . . . .	73
7.4 Tool File . . . . .	74
7.5 Parameters . . . . .	75
7.6 Coordinate Systems . . . . .	75
<b>8 Language Overview</b>	<b>77</b>
8.1 Format of a line . . . . .	77
8.2 Line Number . . . . .	78
8.3 Word . . . . .	78
8.3.1 Number . . . . .	78
8.3.2 Parameter Value . . . . .	79
8.3.3 Expressions and Binary Operations . . . . .	79
8.3.4 Unary Operation Value . . . . .	80
8.4 Parameter Setting . . . . .	80
8.5 Comments and Messages . . . . .	80
8.6 Repeated Items . . . . .	80
8.7 Item order . . . . .	81
8.8 Commands and Machine Modes . . . . .	81
8.9 Modal Groups . . . . .	81
<b>9 G Codes</b>	<b>83</b>
9.1 G0: Rapid Linear Motion . . . . .	83
9.2 G1: Linear Motion at Feed Rate . . . . .	83
9.3 G2, G3: Arc at Feed Rate . . . . .	84
9.3.1 Radius format arcs . . . . .	84
9.3.2 Center format arcs . . . . .	84
9.4 G33: Spindle-Synchronized Motion . . . . .	85
9.5 G4: Dwell . . . . .	86
9.6 G10: Set Coordinate System Data . . . . .	86
9.7 G17, G18, G19: Plane Selection . . . . .	86
9.8 G21, G21: Length Units . . . . .	86
9.9 G28, G30: Return to Home . . . . .	86
9.10 G38.2: Straight Probe . . . . .	87
9.11 G40, G41, G42: Cutter Radius Compensation. . . . .	87

9.12G43, G49: Tool Length Offsets . . . . .	87
9.13G53: Move in absolute coordinates . . . . .	87
9.14G54 to G59.3: Select Coordinate System . . . . .	88
9.15G61, G61.1, G64: Set Path Control Mode . . . . .	88
9.16G80: Cancel Modal Motion . . . . .	88
9.17G81 to G89: Canned Cycles . . . . .	88
9.17.1Preliminary and In-Between Motion . . . . .	89
9.17.2G81: Drilling Cycle . . . . .	90
9.17.3G82: Drilling Cycle with Dwell . . . . .	91
9.17.4G83: Peck Drilling . . . . .	91
9.17.5G84: Right-Hand Tapping . . . . .	91
9.17.6G85: Boring, No Dwell, Feed Out . . . . .	91
9.17.7G86: Boring, Spindle Stop, Rapid Out . . . . .	92
9.17.8G87: Back Boring . . . . .	92
9.17.9G88: Boring, Spindle Stop, Manual Out . . . . .	92
9.17.10G89: Boring, Dwell, Feed Out . . . . .	92
9.17.11G90, G99: Set Distance Mode . . . . .	92
9.18G92, G92.1, G92.2, G92.3: Coordinate System Offsets . . . . .	93
9.19G93, G94: Set Feed Rate Mode . . . . .	93
9.20G98, G99: Set Canned Cycle Return Level . . . . .	94
<b>10M Codes</b>	<b>95</b>
10.1M0, M1, M2, M30, M60: Program Stopping and Ending . . . . .	95
10.2M3, M4, M5: Spindle Control . . . . .	95
10.3M6: Tool Change . . . . .	96
10.4M7, M8, M9: Coolant Control . . . . .	96
10.5M48, M49: Override Control . . . . .	96
10.6M100 to M199: User Defined Commands . . . . .	96
<b>11O Codes</b>	<b>97</b>
11.1Subroutines: “sub”, “endsub”, “return”, “call” . . . . .	97
11.2Looping: “do”, “while”, “endwhile”, “break”, “continue” . . . . .	97
11.3Conditional: “if”, “else”, “endif” . . . . .	98
<b>12Other Codes</b>	<b>99</b>
12.1F: Set Feed Rate . . . . .	99
12.2S: Set Spindle Speed . . . . .	99
12.3T: Select Tool . . . . .	99
<b>13Order of Execution</b>	<b>100</b>
<b>14G-Code Best Practices</b>	<b>101</b>
14.1Use an appropriate decimal precision . . . . .	101
14.2Use consistent white space . . . . .	101
14.3Prefer “Center-format” arcs . . . . .	101
14.4Put important modal settings at the top of the file . . . . .	101
14.5Don’t put too many things on one line . . . . .	101
14.6Don’t use line numbers . . . . .	101

<b>15 Tool File and Compensation</b>	<b>102</b>
15.1 Tool File . . . . .	102
15.2 Tool Compensation . . . . .	103
15.3 Tool Length Offsets . . . . .	103
15.4 Cutter Radius Compensation . . . . .	104
15.4.1 Cutter Radius Compensation Detail . . . . .	104
15.5 Tool Compensation Sources . . . . .	111
<b>16 Coordinate System and G92 Offsets</b>	<b>113</b>
16.1 Introduction . . . . .	113
16.2 The Machine Position Command (G53) . . . . .	113
16.3 Fixture Offsets (G54-G59.3) . . . . .	113
16.3.1 Default coordinate system . . . . .	115
16.3.2 Setting coordinate system values within G-code. . . . .	115
16.4 G92 Offsets . . . . .	115
16.4.1 The G92 commands . . . . .	116
16.4.2 Setting G92 values . . . . .	116
16.4.3 G92 Cautions . . . . .	117
16.5 Sample Program Using Offsets . . . . .	117
<b>A Glossary of Common Terms Used in the EMC Documents</b>	<b>119</b>
<b>A Legal Section</b>	<b>122</b>
A.1 GNU Free Documentation License Version 1.1, March 2000 . . . . .	122
A.1.1 GNU Free Documentation License Version 1.1, March 2000 . . . . .	122



## **Part I**

# **Introduction & installing EMC2**

# Chapter 1

## The Enhanced Machine Control

### 1.1 Introduction

This book is intended for people who want to use the Enhanced Machine Controller to run a mill, lathe, router, or to control some other rather standard kind of machine. Computer Numerical Control or CNC is the general term used to name this kind of computer application. In order to get right into the essential task of operating it we have limited the amount of information about installation and setup. We assume that the user will install from one of the standard ways of install (covered in Chapter 2). Machine wiring and setup is limited to what we refer to as a mini or benchtop mill that is powered by stepper motors and amps that use a single parallel port.

If the user is interested in developing their own install using some other distribution of Linux or an other operating system, or applying the EMC2 to a more complex machine, they should study the Integrators Handbook where these topics are covered in greater detail.

### 1.2 The Big CNC Picture

The term CNC has taken on a lot of different meanings over the years. In the early days it replaced the hands of a skilled machinist with motors that followed commands in much the same way that the machinist turned the handwheels. From these early machines, a language of machine tool control has grown. This language is called RS274 and several standard variants of it have been put forward. It has also been expanded by machine tool and control builders in order to meet the needs of specific machines. If a machine changed tools during a program it needed to have tool change commands. If it changed pallets in order to load new castings, it had to have commands that allowed for these kinds of devices as well. Like any language, RS274 has evolved over time. Currently there are several dialects. In general each machine tool maker has been consistent within their product line but different dialects can have commands that cause quite different behavior from one machine to another.

More recently the language of CNC has been hidden behind or side-stepped by several programming schemes that are referred to as “Conversational<sup>1</sup> programming languages.” One common feature of these kinds of programming schemes is the selection of a shape or geometry and the addition of values for the corners, limits, or features of that geometry.

The use of Computer Aided Drafting has also had its affect upon the CNC programming languages. Because CAD drawings are saved as a list or database of geometries and variables associated with each, they are available to be interpreted into G-Code. These interpreters are called CAM (Computer Aided Machining) programs.

Like the CAD converters, the rise of drawing programs, like Corel<sup>TM</sup> and the whole bunch of paint programs, converters have been written that will take a bitmap or raster or vector image and turn

---

<sup>1</sup>One machine tool manufacturer, Hurco, claims to have a right to the use of these programming schemes and to the use of the term conversational when used in this context.

it into G-Code that can be run with a CNC.

You're asking yourself, "Why did I want to know this?" The answer is that the EMC2 as is currently exists does not directly take in CAD or any image and run a machine using it. It. The EMC2 uses a variant of the earlier CNC language named RS 274NGC. (Next Generation Controller). All of the commands given to the EMC2 must be in a form that is recognized and have meaning to the RS274NGC interpreter. This means that if you want to carve parts that were drawn in some graphical or drafting program you will also have to find a converter that will transform the image or geometry list into commands that are acceptable to the EMC2 interpreter. Several commercial CAD/CAM programs are available to do this conversion. At least one converter (Ace) has been written that carries a copyright that makes it available to the public.

There has been recent talk about writing a "conversational" or geometric interface that would allow an operator to enter programs in much the same way that several modern proprietary controls enter programs but it isn't in there yet.

### 1.3 Computer Operating Systems

The EMC2 code can be compiled on almost any GNU-Linux Distribution (assuming it has been patched with a real time extension). In addition to the raw code, some binary distributions are available. The latest packages have been created around the Ubuntu GNU-Linux Distribution. Ubuntu is one of the distributions that is aimed at novice Linux users, and has been found to be very easy to use. Along with that, there are lots of places around the world, that offer support for it. Installing emc2 on it is trivial, as you can see from Chapter 2.

The EMC2 will not run under a normal Microsoft (TM) operating system. The reason for this is that the EMC2 requires a real-time environment for the proper operation of its motion planning and stepper pulse outputs. Along with that, it also benefits from the much-needed stability and performance of the Linux OS.

### 1.4 History of the Software

The EMC2 code was started by the Intelligent Systems Division at the National Institute of Standards and Technology in the United States. The quotation below, taken from the NIST web presence some time back, should lend some understanding of the essential reasons for the existence of this software and of the NIST involvement in it.

As part of our (NIST) collaboration with the OMAC User's Group, we have written software which implements real-time control of equipment such as machine tools, robots, and coordinate measuring machines. The goal of this software development is twofold: first, to provide complete software implementations of all OMAC modules for the purpose of validating application programming interfaces; and second, to provide a vehicle for the transfer of control technology to small- and medium-sized manufacturers via the NIST Manufacturing Extension Partnership. The EMC2 software is based on the NIST Real-time Control System (RCS) Methodology, and is programmed using the NIST RCS Library. The RCS Library eases the porting of controller code to a variety of Unix and Microsoft platforms, providing a neutral application programming interface (API) to operating system resources such as shared memory, semaphores, and timers. The RCS Library also implements a communication model, the Neutral Manufacturing Language, which allows control processes to read and write C++ data structures throughout a single homogeneous environment or a heterogeneous networked environment. The EMC2 software is written in C and C++, and has been ported to the PC Linux, Windows NT, and Sun Solaris operating systems. When running actual equipment, a real-time version of Linux is used to achieve the deterministic computation rates required (200 microseconds is typical). The software can also be run entirely in simulation, down to simulations of the machine motors. This enables entire factories of EMC2 machines to be set up and run in a computer integrated manufacturing environment.

EMC has been installed on many machines, both with servo motors and stepper motors. Here is a sampling of the earliest applications.

- 3-axis Bridgeport knee mill at Shaver Engineering. The machine uses DC brush servo motors and encoders for motion control, and OPTO-22 compatible I/O interfaced to the PC parallel port for digital I/O to the spindle, coolant, lube, and e-stop systems.
- 3-axis desktop milling machine used for prototype development. The machine uses DC brush servo motors and encoders. Spindle control is accomplished using the 4th motion control axis. The machine cuts wax parts.
- 4-axis Kearney & Trecker horizontal machining center at General Motors Powertrain in Pontiac, MI. This machine ran a precursor to the full-software EMC2 which used a hardware motion control board.

After these early tests, Jon Elson found the Shaver Engineering notes and replaced a refrigerator sized Allen Bradley 7300 control on his Bridgeport with the EMC running on a Red Hat 5.2 distribution of Linux. He was so pleased with the result that he advertised the software on several news groups. He continues to use that installation and has produced several boards that are supported by the software.

From these early applications news of the software spread around the world. It is now used to control many different kinds of machines. More recently the Sherline company <http://www.sherline.com> has released their first CNC mill. It uses a standard release of the EMC.

The source code files that make up the controller are kept in a repository on [cvs.linuxcnc.org](http://cvs.linuxcnc.org) They are available for anyone to download and use as they see fit. All EMC2 files (with some exceptions<sup>2</sup>) carry a GPL or GPLD copyright. These files require that you make your source of them available to your users. GPL copyright also requires that if you modify the code in that file and make a public release of your revisions, you must return your modifications to the developers.

## 1.5 How the EMC2 Works

The Enhanced Machine Controller (EMC2) is a lot more than just another CNC mill program. It can control machine tools, robots, or other automated devices. It can control servo motors, stepper motors, relays, and other devices related to machine tools. In this handbook we focus on only a small part of that awesome capability, the minimill.

Figure 1.1 shows a simple block diagram showing what a typical 3-axis EMC2 system might look like. This diagram shows a stepper motor system. The PC, running Linux as its operating system, is actually controlling the stepper motor drives by sending signals through the printer port. These signals (pulses) make the stepper drives move the stepper motors. The EMC2 can also run servo motors via servo interface cards or by using an extended parallel port to connect with external control boards. As we examine each of the components that make up an EMC2 we will remind the reader of this typical machine.

There are four main components to the EMC2 software: a motion controller (EMCMOT), a discrete I/O controller (EMCIO), a task executor which coordinates them (EMCTASK), and a collection of text-based or graphical user interfaces. An EMC2 capable of running a minimill must start some version of all four of these components in order to completely control it. Each component is briefly described below. In addition there is a layer called HAL (Hardware Abstraction Layer) which allows simple reconfiguration of EMC2 without the need of recompiling.

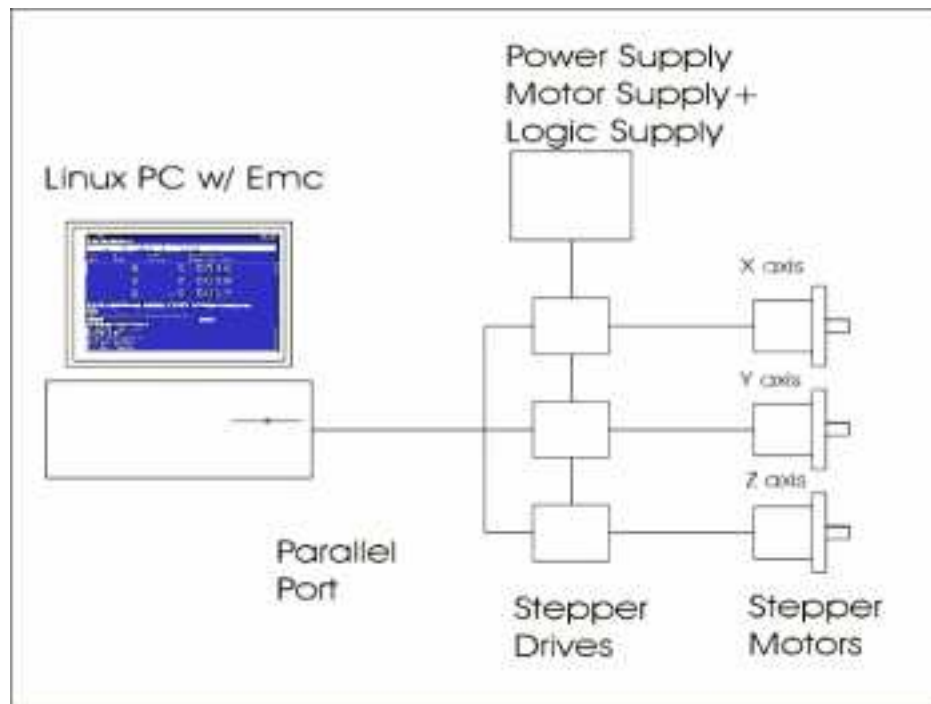
### 1.5.1 Graphical User Interfaces

A graphical interface is the part of the EMC2 that the machine tool operator interacts with. The EMC2 comes with several types of user interfaces:

---

<sup>2</sup>some parts of emc2 are released under LGPL, to allow proprietary software to be linked together with emc2 (GUIs, hardware drivers, etc.)

Figure 1.1: Typical EMC2 Controlled Machine



- an interactive command-line program named `emcpanel`
- a character-based screen graphics program named `keystick` 1.3
- X Windows programs named `xemc` 1.6 and `yemc`
- a Java-based GUI, `emcgui`
- two Tcl/Tk-based GUIs named `tkemc` 1.5 and `mini` 1.4.
- a modern GL-based GUI written in python called `AXIS` 1.2<sup>3</sup>

`Tkemc` and `Mini` are most commonly used operator interfaces. They will run on Linux, Mac, and Microsoft Windows if the Tcl/Tk programming language has been installed. The Mac and Microsoft Windows version can connect to a real-time EMC2 running on a Linux machine via a network connection, allowing the monitoring of the machine from a remote location. Instructions for installing and configuring the connection between a Mac or Microsoft Machine and a PC running the EMC2 can be found in the integrators handbook.

### 1.5.2 Motion Controller EMC2MOT

Motion control includes sampling the position of the axes to be controlled, computing the next point on the trajectory, interpolating between these trajectory points, and computing an output to the motors. For servo systems, the output is based on a PID compensation algorithm. For stepper systems, the calculations run open-loop, and pulses are sent to the steppers based on whether their accumulated position is more than a pulse away from where their commanded position should be. The motion controller includes programmable software limits, interfaces to hardware limit and home switches.

<sup>3</sup>AXIS has been developed outside the EMC2 project, you can find information about it at <http://axis.unpy.net>

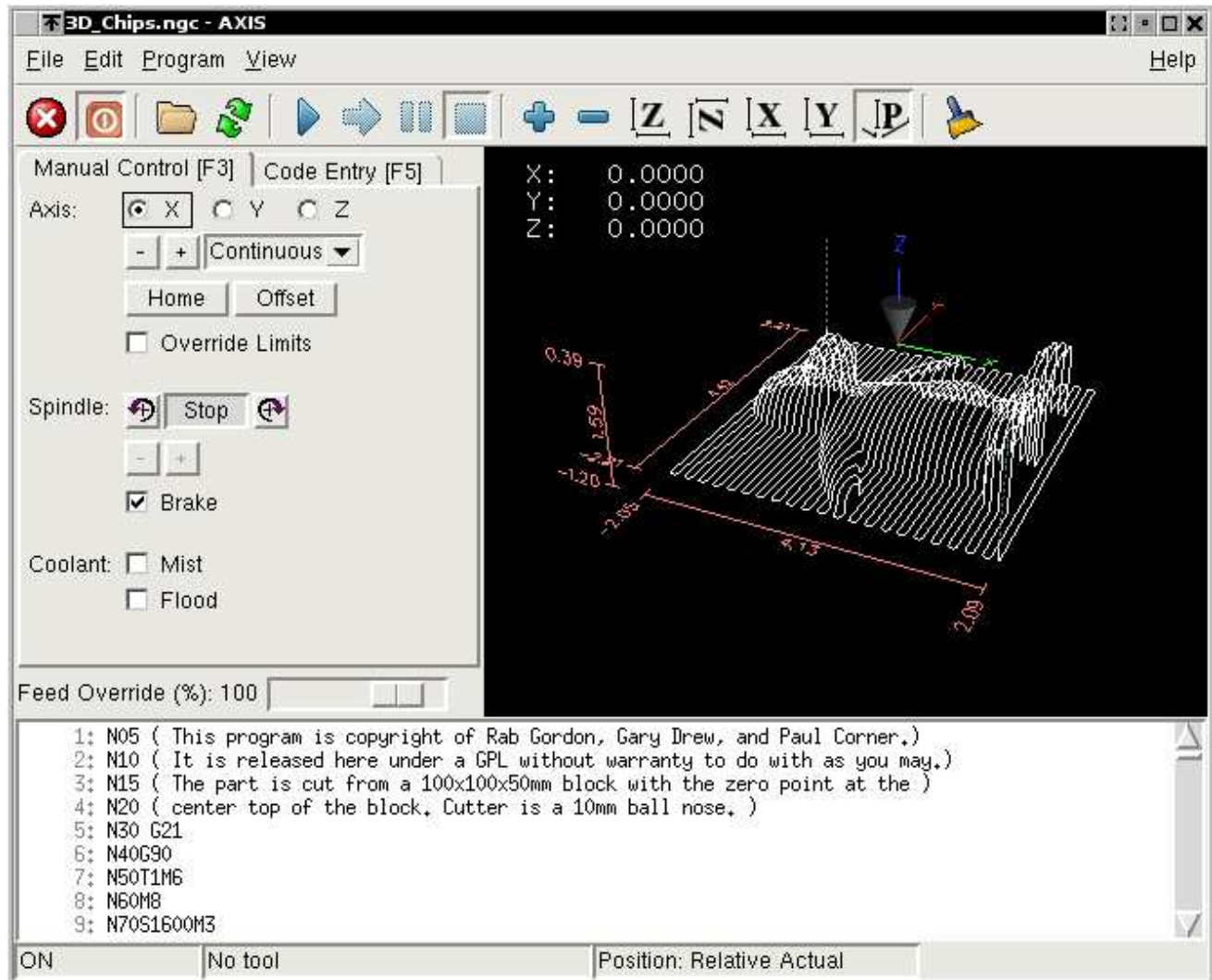


Figure 1.2: The AXIS Graphical Interface

The motion controller is written to be fairly generic. Initialization files (with the same syntax as Microsoft Windows INI files) are used to configure parameters such as number and type of axes (e.g., linear or rotary), scale factors between feedback devices (e.g., encoder counts) and axis units (e.g., millimeters), servo gains, servo and trajectory planning cycle times, and other system parameters. Complex kinematics for robots can be coded in C according to a prescribed function interface and linked in to replace the default 3-axis Cartesian machine kinematics routines.

The motion controllers that you will be using with your stepper motors will most likely be stepgen. The ability and requirements of each of these will be described when we get to hardware and how to configure the EMC2 for your specific hardware.

### 1.5.3 Discrete I/O Controller EMCIO

Discrete I/O controllers are highly machine-specific, and are not customizable in general using the INI file technique used to configure the more generic motion controller. However, since EMC2 uses the HAL, reconfiguration of the IO subsystem has become very powerful and flexible. EMC2 contains even a Programmable Logic Controller module (behaves just like a hardware PLC), which can be used for very complex scenarios (tool changers, etc.).

In EMC2 there is only one big I/O controller, which provides support for all kinds of actions and

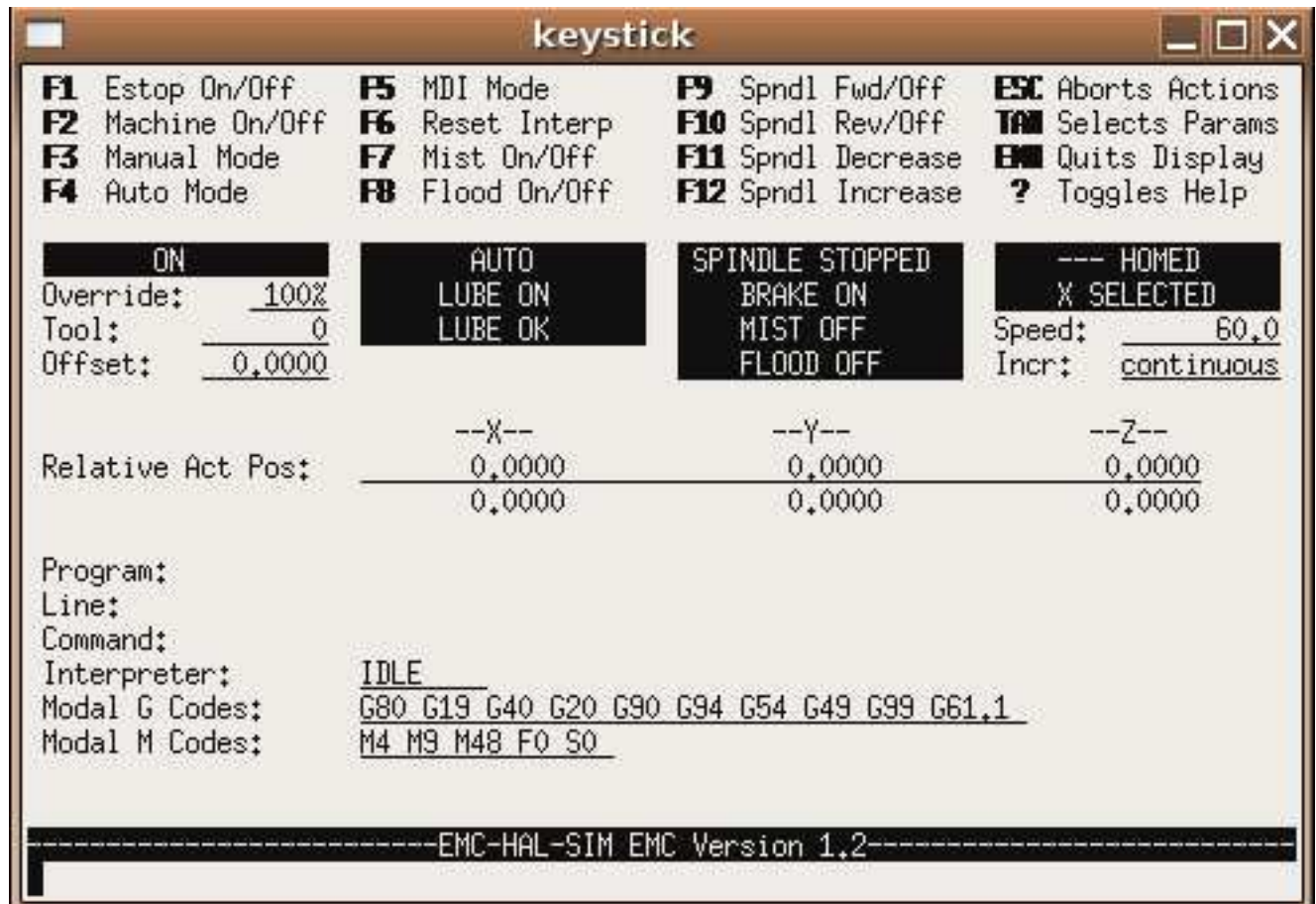


Figure 1.3: The Keystick interface

hardware control. All its outputs and inputs are HAL logic items (more on this later on), so you can use only the subset which interests you (or the one that fist on your hardware output: e.g. if you have only one parallel port, and no additional I/O cards).

#### 1.5.4 Task Executor EMCTASK

The Task Executor is responsible for interpreting G and M code programs whose behavior does not vary appreciably between machines. G-code programming is designed to work like a machinist might work. The motion or turns of a handwheel are coded into blocks. If a machinist wanted his mill to move an inch in the +X direction at some feedrate, he might slowly turn the handwheel five turns clockwise in 20 seconds. The same machinist programming that same move for CNC might write the following block of code.

```
G1 F3 X1.000
```

G1 means that the machine is supposed to run at a programmed feedrate rather than at the fastest speed that it can. (G0) The F3 means that it should travel at 3 inches a minute or 3 millimeters a minute if it is working in metric mode. The X1.000 assumes that the machine started at zero and is supposed to go one inch in the positive direction. You will read quite a bit more about G-code in the programming chapters.

Figure 1.7 is a block diagram of how a personal computer running the EMC2 is used to control a machine with G-code. The actual G-code can be sent using the MDI (Machine Device Interface)

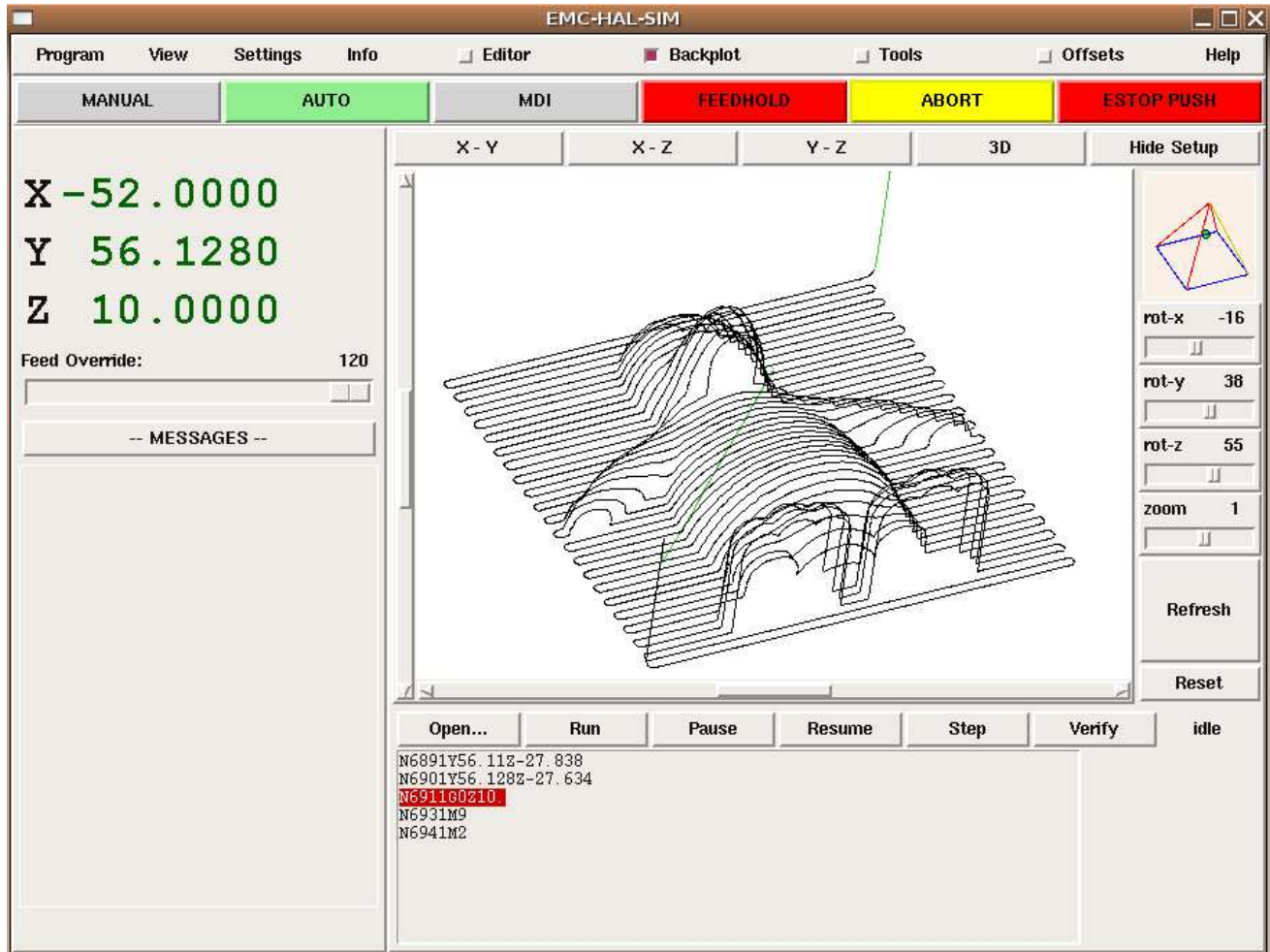


Figure 1.4: The Mini Graphical Interface

mode or it can be sent as a file when the machine is in Auto mode. These choices are made by the operator and entered using one of the Graphical User Interfaces available with the software.

G-code is sent to the interpreter which compares the new block with what has already been sent to it. The interpreter then figures out what needs to be done for the motion and input or output systems and sends blocks of canonical commands to the task and motion planning programs.

## 1.6 Thinking Like a Machine Operator

This book will not even pretend that it can teach you to run a mill or a lathe. Becoming a machinist takes time and hard work. An author once said, “We learn from experience, if at all.” Broken tools, gouged vices, and scars are the evidence of lessons taught. Good part finish, close tolerances, and careful work are the evidence of lessons learned. No machine, no computer program, can take the place of human experience.

As you begin to work with the EMC2 program, you will need to place yourself in the position of operator. You need to think of yourself in the role of the one in charge of a machine. It is a machine that is either waiting for your command or executing the command that you have just given it. Throughout these pages we will give information that will help you become a good operator of the EMC2 mill. You will need some information right up front here so that the following pages will make sense to you.



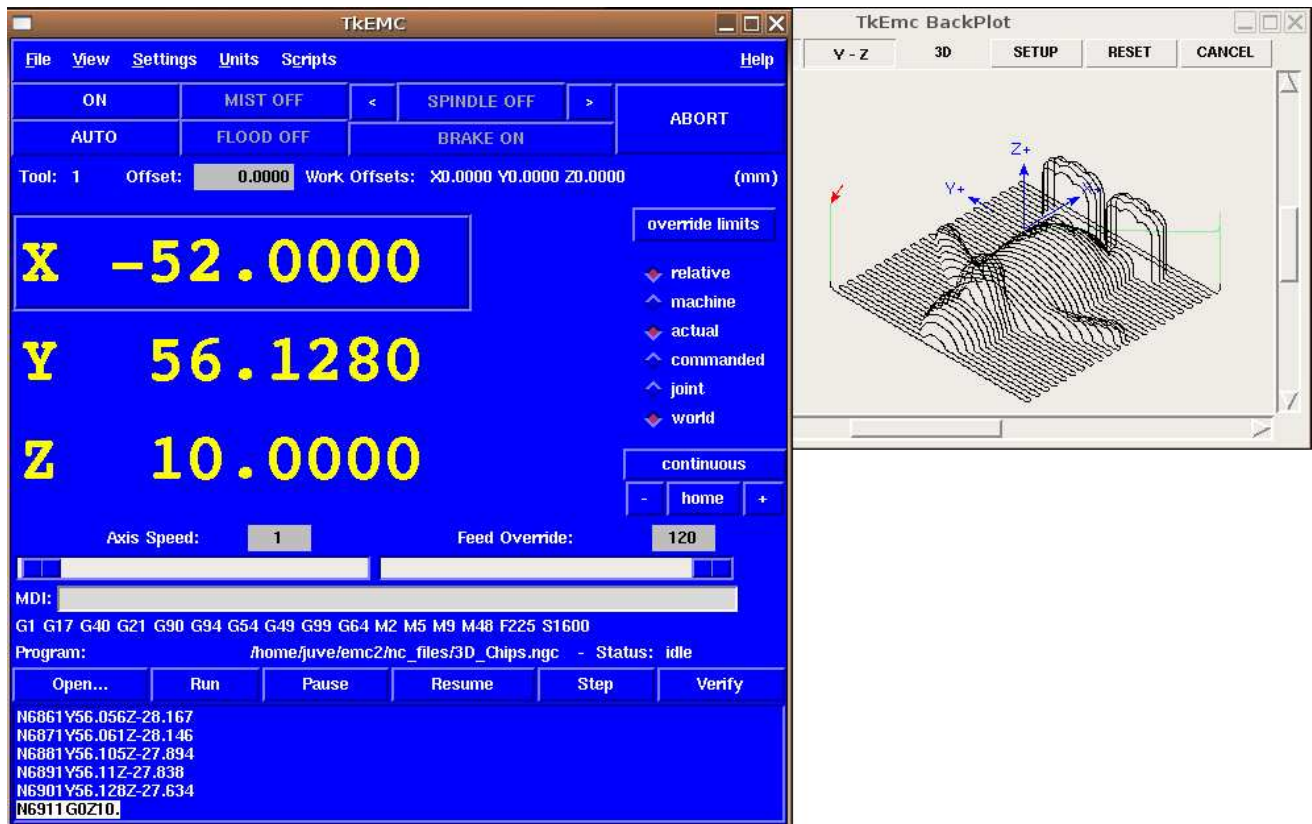


Figure 1.5: The TkEmc Graphical Interface

### 1.6.1 Modes of Operation

When an EMC2 is running, there are three different major modes used for inputting commands. These are Manual, Auto, and MDI. Changing from one mode to another makes a big difference in the way that the EMC2 behaves. There are specific things that can be done in one mode that can not be done in another. An operator can home an axis in manual mode but not in auto or MDI modes. An operator can cause the machine to execute a whole file full of G-codes in the auto mode but not in manual or MDI.

In manual mode, each command is entered separate. In human terms a manual command might be “turn on coolant” or “jog X at 25 inches per minute.” These are roughly equivalent to flipping a switch or turning the handwheel for an axis. These commands are normally handled on one of the graphical interfaces by pressing a button with the mouse or holding down a key on the keyboard. In auto mode, a similar button or key press might be used to load or start the running of a whole program of G-code that is stored in a file. In the MDI mode the operator might type in a block of code and tell the machine to execute it by pressing the <return> or <enter> key on the keyboard.

Some motion control commands are available and will cause the same changes in motion in all modes. These include ABORT, ESTOP, and FEEDRATE OVERRIDE. Commands like these should be self explanatory.

### 1.6.2 Information Display

While an EMC2 is running, each of the modules keeps up a dialog with each other and with the graphical display. It is up to the display to select from that stream of information what the operator needs to see and to arrange it on the screen in a way that makes it easy for the operator to understand. Perhaps the most important display is the mode the EMC2 is running in. You will want to

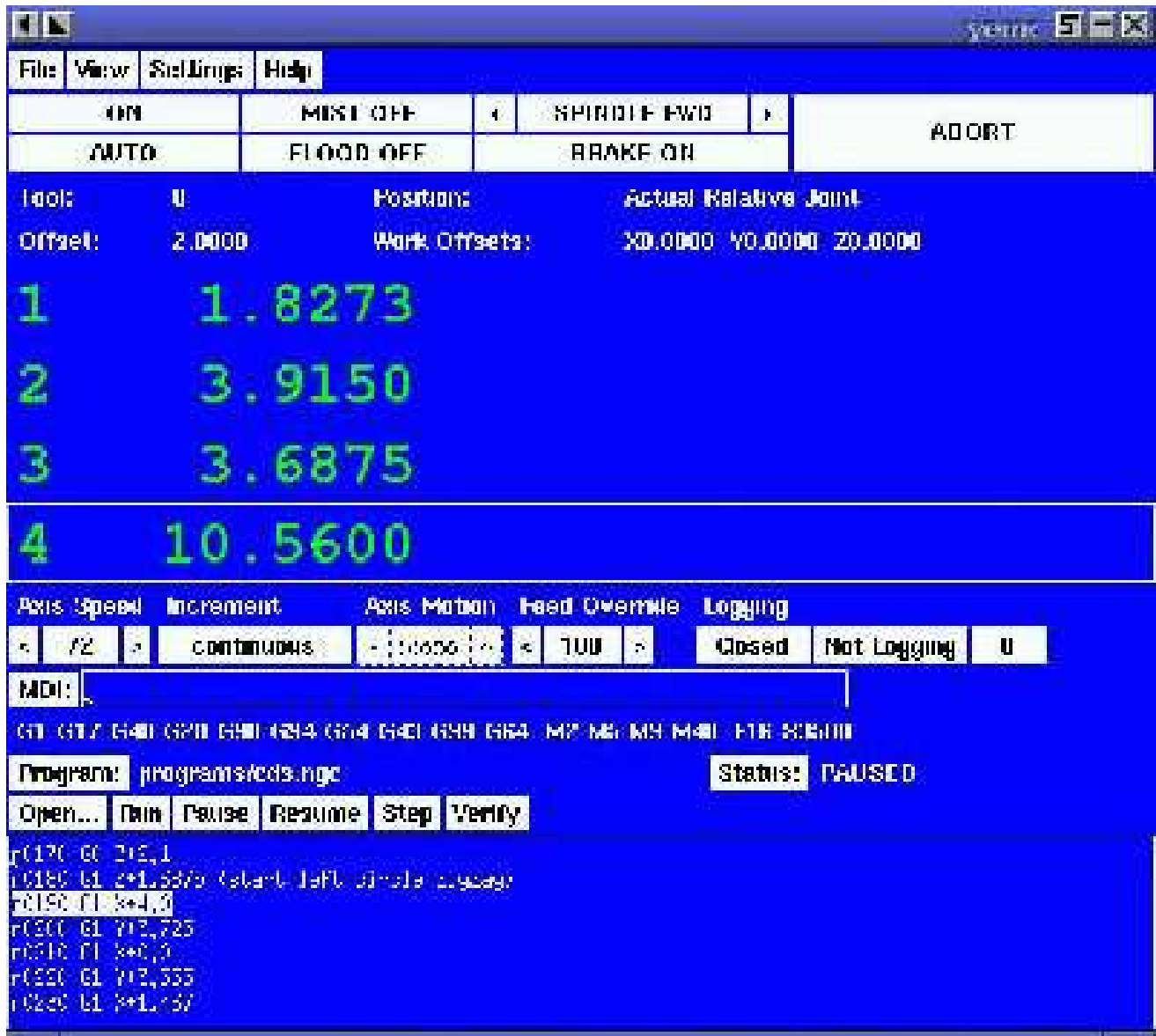


Figure 1.6: The XEMC Graphical Interface

keep your eye on the mode display.

Right up there with knowing what mode is active is consistent display of the position of each axis. Most of the interfaces will allow the operator to read position based upon actual or commanded position as well as machine or relative position.

**Machine** This is the position of an axis relative to the place where it started or was homed.

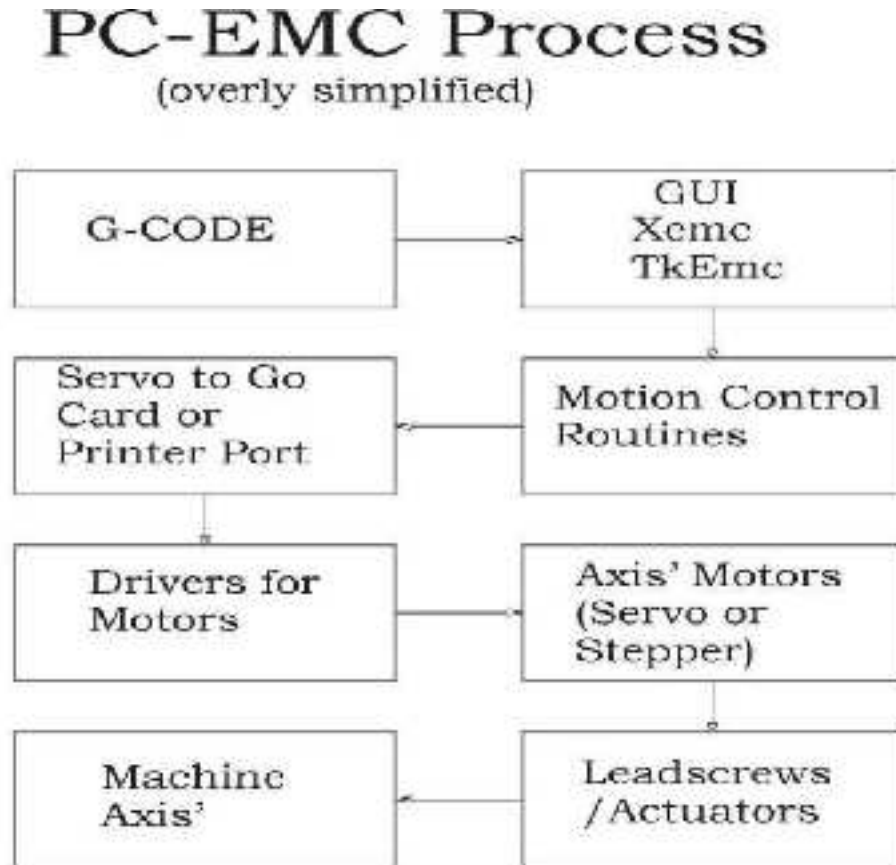
**Relative** This is the position of an axis after work or tool or other offsets have been applied.

**Actual** This is the real position of the axis within the machine or relative system.

**Commanded** This is where the axis is commanded to be.

These may all be exactly the same if no offsets have been applied and there is no deadband set in the INI file. Deadband is a small distance which is assumed to be close enough – perhaps one stepper pulse or one encoder pulse.

Figure 1.7: EMC2 Process Diagram



It is also important to see any messages or error codes sent by the EMC2. These are used by operators if they need to be reminded to change a tool, to clear up problems in G-code programs, or to find out why the machine stopped running.

As you work your way through this text, you will be learning, bit by bit how to setup and run a machine with your copy of the EMC2 software. While you are learning about setting up and running a minimill here, you will be thinking of other applications and other capabilities. These are the topics of the other linuxcnc.org handbooks.

## 1.7 Thinking Like An Integrator

The biggest task of a machine integrator is figuring out how to connect a PC running the EMC2 to a machine and configuring the software so that it runs the machine correctly. Most of this is not the topic of this book but there are a few things that you will have to understand in order to make our little minimill work for us like we expect it to work.

### 1.7.1 Units

Units can be confusing. A newbie recently asked, “Does it work in inches, feet, centimeters, millimeters, or what?” There are several possible answers to this question but the best one is that it works in the units that you set it to work in.

At a machine level, we set each axis’ units to some value using an INI variable that looks like this.

```
UNITS = 1
```

or

```
UNITS = 0.03937007874016
```

Math folk will get a clue from these numbers because the long decimal number is the distance represented by one millimeter if we convert it into inches. “So,” you say, “the EMC2 uses millimeters internally.” If we use `UNITS = 1` then we have defined our user units as millimeters. If we use `UNITS = 0.03937007874016` then we have defined our user units as inches. Using similar arithmetic we could set our units to most any value we wanted. (Some of the EMC2 people who run vehicles with the EMC2 set units to kilometers or miles.)

After we have decided upon a value for the units for an axis, we tell the EMC2 how many step pulses or encoder pulses it should send or read for each unit of distance to be traveled. Once we have done this, the EMC2 knows how to count units of distance. However it is very important to understand that this counting of distance is different from the commanding of distance. You can command distance in millimeters or inches without even thinking about the units that you defined. There are G-codes that allow you to switch easily between metric and imperial.

### 1.7.2 Some things we may not want to change.

Within the EMC2 code are a few things that are not easily changed. We call these defaults. There are connections that have been made between the running components of the EMC2 that we can not easily change. We’ll see that there are displays and buttons and keyboard keys that are not easily shifted about. We’ll learn about and get used to these in the chapters ahead.

### 1.7.3 Some things we will need to change.

The EMC2 is configured with files that are read at startup and used to override the compiled defaults. No real controller will likely use the compiled defaults, so you will certainly need to edit at least some of these files to reflect the specifics of your machine.

There are five kinds of configuration files: INI, NML, TBL, VAR and HAL files. These are reflected in lower case file extensions to a file name. They may be named `EMC2.tbl` or `generic.tbl` but they do the same thing when they are read by the EMC2 as it starts up. Many users copy these and name them for the specific machine that they are editing them for. A set of these files named `Sherlinemill.ini`, `Sherlinemill.var`, `Sherlinemill.tbl` and `Sherlinemill.nml` are certainly more descriptive than a bunch of files named `generic`.

These files each contain specific information for your CNC.

- `stepper.ini`, contains all the machine parameters such as servo gains, scale factors, cycle times, units, etc. and will certainly need to be edited.
- `emc.nml` contains communication settings for shared memory and network ports you may need to override on your system, although it is likely that you can leave these settings alone.
- `stepper.tbl` contains the tool information such as which pocket contains which tool, and the length and diameter for each tool.
- `rs274ngc.var` contains variables specific to the RS-274-NGC dialect of NC code, notably for setting the persistent numeric variables for the nine work coordinate systems.

We’ll get into some of the details of these files as we begin to hook up and operate our little machine. In addition to these four files, there is a standard startup file. Back in the early days of the EMC it was common to have to start up several different tasks in different terminal windows in order to get the EMC to run a machine. Each of these tasks had to be supplied a bunch of information in the

form of arguments in order to be certain that the task started the way that we expected it to. All of this was a tedious thing. All of this has been replaced with the run script file. It is named simply 'emc'. This executable script file controls the startup of all of the modules needed to run a standard version of the EMC2. When run, it lets the user chose a certain config to run EMC2 from.

## Chapter 2

# Installing the EMC2 software

### 2.1 Introduction

One of the problems users often complained about EMC was installing the software itself. They were forced to get sources, and compile themselves, and try to set up a RT-patched Linux, etc. The developers of EMC2 chose to go with a standard distribution called Ubuntu<sup>1</sup>.

Ubuntu has been chosen, because it fits perfectly into the Open Source views of EMC2:

- Ubuntu will always be free of charge, and there is no extra fee for the "enterprise edition", we make our very best work available to everyone on the same Free terms.
- Ubuntu comes with full professional support on commercial terms from hundreds of companies around the world, if you need those services. Each new version of Ubuntu receives free security updates for 18 months after release, some versions are supported for even longer.
- Ubuntu uses the very best in translations and accessibility infrastructure that the Free Software community has to offer, to make Ubuntu usable for as many people as possible.
- Ubuntu is released regularly and predictably; a new release is made every six months. You can use the current stable release or help improve the current development release.
- The Ubuntu community is entirely committed to the principles of free software development; we encourage people to use open source software, improve it and pass it on.

### 2.2 EMC Download Page

You will find the most recent releases of EMC2 announced on [www.linuxcnc.org](http://www.linuxcnc.org). The releases of EMC2 will be done in two ways (sources and binary package). The sources (described in the Developers Handbook) consist of a tarball (`emc2-version.tar.gz`), which you should download and unpack into your home directory.

This document (oriented towards the end-user) will only try to explain how to install the binary package on the Ubuntu distribution<sup>2</sup>.

---

<sup>1</sup>"Ubuntu" is an ancient African word, meaning "humanity to others". Ubuntu also means "I am what I am because of who we all are". The Ubuntu Linux distribution brings the spirit of Ubuntu to the software world. You can read more about it at <http://www.ubuntu.com>

<sup>2</sup>For information regarding other Linux variants, check the Developers Handbook or ask for help on the emc-developers mailing list [http://sourceforge.net/mail/?group\\_id=6744](http://sourceforge.net/mail/?group_id=6744).

## 2.3 EMC2 install script - the easy way to install

Chris Radek put together a simple script to install emc2 on Ubuntu. It runs the commands explained in 2.4.

To use it you need to :

- Download the script from <http://www.linuxcnc.org/emc2-install.sh>
- Save it on your Desktop. Right-click the icon, select Properties. Go to the Permissions tab and check the box for Owner: Execute. Close the Properties window.
- Now double-click the emc2-install.sh icon, and select "Run in Terminal". A terminal will appear and you will be asked for your password.
- When the installation asks if you are sure you want to install the EMC2 packages, hit Enter to accept. Now just allow the install to finish.
- When it is done, you must reboot (System > Log Out > Restart the Computer), and when you log in again you can run EMC2 by selecting it on the Applications > Other menu.
- If you aren't ready to set up a machine configuration, try the sim-AXIS configuration; it runs a "simulated machine" that requires no attached hardware.
- Now that the initial installation is done, Ubuntu will prompt you when updates of EMC2 or its supporting files are available. When they are, you can update them easily and automatically with the Update Manager.

## 2.4 Manual installing using apt commands.

The following few section will describe how to install EMC2 using a console and apt-commands. If you know a bit about Linux and Debian-flavored distributions this might be trivial. If not, you might consider reading 2.3.

First add the repository to /etc/apt/sources.list:

```
$ sudo sh -c 'echo "deb http://dsplabs.cs.upt.ro/emc2/ breezy emc2" >>/etc/apt/sources.list; echo "deb-src http://dsplabs.c
```

Then update & get emc2-axis.

```
$ sudo apt-get update
$ sudo apt-get install emc2-axis
```

This command will install the emc2-axis<sup>3</sup> package along with all dependencies<sup>4</sup>.

You might get warnings that the packages are from an untrusted source (this means your computer doesn't recognize the GPG signature on the packages). To correct that issue the following commands:

```
$ gpg --keyserver pgpkeys.mit.edu --recv-key BC92B87F
$ gpg -a --export BC92B87F | sudo apt-key add -
```

<sup>3</sup>The emc2-axis package is the AXIS gui packaged for emc2.

<sup>4</sup>The dependencies are one of the nicest thing in Debian based distributions. They assure you have everything installed that you need. In the case of emc2 it's even a RT-patched kernel, and all needed libraries.

**Part II**

**Configuring EMC2**





# Chapter 3

## Introduction

### 3.1 What is HAL?

HAL stands for Hardware Abstraction Layer. At the highest level, it is simply a way to allow a number of “building blocks” to be loaded and interconnected to assemble a complicated system. The “Hardware” part is because HAL was originally designed to make it easier to configure EMC for a wide variety of hardware devices. Many of the building blocks are drivers for hardware devices.

However, HAL can do more than just configure hardware drivers.

#### 3.1.1 HAL is based on traditional system design techniques

HAL is based on the same principles that are used to design hardware circuits and systems, so it is useful to examine those principles first.

Any system (including a CNC machine), consists of interconnected components. For the CNC machine, those components might be the main controller, servo amps or stepper drives, motors, encoders, limit switches, pushbutton pendants, perhaps a VFD for the spindle drive, a PLC to run a toolchanger, etc. The machine builder must select, mount and wire these pieces together to make a complete system.

##### 3.1.1.1 Part Selection

The machine builder does not need to worry how each individual part works. He treats them as black boxes. During the design stage, he decides which parts he is going to use - steppers or servos, which brand of servo amp, what kind of limit switches and how many, etc. The integrator's decisions about which specific components to use is based on what that component does and the specifications supplied by the manufacturer of the device. The size of a motor and the load it must drive will affect the choice of amplifier needed to run it. The choice of amplifier may affect the kinds of feedback needed by the amp and the velocity or position signals that must be sent to the amp from a control.

In the HAL world, the integrator must decide what HAL components are needed. Usually every interface card will require a driver. Additional components may be needed for software generation of step pulses, PLC functionality, and a wide variety of other tasks.

##### 3.1.1.2 Interconnection Design

The designer of a hardware system not only selects the parts, he also decides how those parts will be interconnected. Each black box has terminals, perhaps only two for a simple switch, or dozens for a servo drive or PLC. They need to be wired together. The motors get connected to the servo amps. The limit switches connect to the controller, and so on. As the machine builder works on the design, he creates a large wiring diagram that shows how all the parts should be interconnected.

When using HAL, components are interconnected by signals. The designer must decide which signals are needed, and what they should connect.

### 3.1.1.3 Implementation

Once the wiring diagram is complete it is time to build the machine. The pieces need to be acquired and mounted, and then they are interconnected according to the wiring diagram. In a physical system, each interconnection is a piece of wire, that needs to be cut and connected to the appropriate terminals.

HAL provides a number of tools to help “build” a HAL system. Some of the tools allow you to “connect” (or disconnect) a single “wire”. Other tools allow you to save a complete list of all the parts, wires, and other information about the system, so that it can be “rebuilt” with a single command.

### 3.1.1.4 Testing

Very few machines work right the first time. While testing the builder may use a meter to see if a limit switch is working, or to measure the DC voltage going to a servo motor. He may hook up an oscilloscope to check the tuning of a drive, or to look for electrical noise. He may find a problem that requires the wiring diagram to be changed - perhaps a part needs to be connected differently or replaced with something completely different.

HAL provides the software equivalent of a voltmeter, oscilloscope, signal generator, and other tools for testing and tuning a system. The same commands used to build the system can be used to make changes as needed.

## 3.1.2 Summary

This document is aimed at people who already know how to do this kind of hardware system integration, but who do not know how to connect the hardware to EMC.

The traditional hardware design as described above ends at the edge of the main control. Outside the control are a bunch of relatively simple boxes, connected together to do whatever is needed.

Inside, the control is a big mystery – one huge black box that we hope works.

HAL extends this traditional hardware design method to the inside of the big black box. It makes device drivers and even some internal parts of the controller into smaller black boxes, that can be interconnected and even replaced just like the external hardware. It allows the "system wiring diagram" to show part of the internal controller, rather than just a big black box. And most importantly it allows the integrator to test and modify the controller using the same methods he would use on the rest of the hardware.

Terms like motors, amps, and encoders are familiar to most machine integrators. When we talk about using extra flexible eight conductor shielded cable to connect an encoder to the servo input board in the computer, the reader immediately understands what it is and is led to the question, “what kinds of connectors will I need to make up each end.” The same sort of thinking is essential for the HAL but the specific train of thought may take a bit to get on track. Using HAL words may seem a bit strange at first, but the concept of working from one connection to the next is the same. This idea of extending the wiring diagram to the inside of the controller is what HAL is all about. If you are comfortable with the idea of interconnecting hardware black boxes, you will probably have little trouble using HAL to interconnect software black boxes.

## 3.2 HAL Concepts

This section is a glossary that defines key HAL terms but it is a bit different than a traditional glossary because these terms are not arranged in alphabetical order. They are arranged by their relationship or flow in the HAL way of things.

**Component:** When we talked about hardware design, we referred to the individual pieces as "parts", "building blocks", "black boxes", etc. The HAL equivalent is a "component" or "HAL component". (This document uses "HAL component" when there is likely to be confusion with other kinds of components, but normally just uses "component".) A HAL component is a piece of software with well defined inputs, outputs, and behaviour, that can be installed and interconnected as needed.

**Parameter:** Many hardware components have adjustments that are not connected to any other components but still need to be accessed. For example, servo amps often have trim pots to allow for tuning adjustments, and test points where a meter or scope can be attached to view the tuning results. HAL components also can have such items, which are referred to as "parameters". There are two types of parameters. Input parameters are equivalent to trim pots - they are values that can be adjusted by the user, and remain fixed once they are set. Output parameters cannot be adjusted by the user - they are equivalent to test points that allow internal signals to be monitored.

**Pin:** Hardware components have terminals which are used to interconnect them. The HAL equivalent is a "pin" or "HAL pin". ("HAL pin" is used when needed to avoid confusion.) All HAL pins are named, and the pin names are used when interconnecting them. HAL pins are software entities that exist only inside the computer.

**Physical\_Pin:** Many I/O devices have real physical pins or terminals that connect to external hardware, for example the pins of a parallel port connector. To avoid confusion, these are referred to as "physical pins". These are the things that "stick out" into the real world.

**Signal:** In a physical machine, the terminals of real hardware components are interconnected by wires. The HAL equivalent of a wire is a "signal" or "HAL signal". HAL signals connect HAL pins together as required by the machine builder. HAL signals can be disconnected and reconnected at will (even while the machine is running).

**Type:** When using real hardware, you would not connect a 24 volt relay output to the +/-10V analog input of a servo amp. HAL pins have the same restrictions, which are based upon their type. Both pins and signals have types, and signals can only be connected to pins of the same type. Currently there are 8 types<sup>1</sup>, as follows:

- BIT - a single TRUE/FALSE or ON/OFF value
- FLOAT - a 32 bit floating point value, with approximately 24 bits of resolution and over 200 bits of dynamic range.
- U8 - an 8 bit unsigned integer, legal values are 0 to +255
- S8 - an 8 bit signed integer, legal values are -128 to +127
- U16 - a 16 bit unsigned integer, legal values are 0 to +65535
- S16 - a 16 bit signed integer, legal values are -32768 to +32767
- U32 - a 32 bit unsigned integer, legal values are 0 to +4294967295
- S32 - a 32 bit signed integer, legal values are -2147483648 to +2147483647

**Function:** Real hardware components tend to act immediately on their inputs. For example, if the input voltage to a servo amp changes, the output also changes automatically. However software components cannot act "automatically". Each component has specific code that must be executed to do whatever that component is supposed to do. In some cases, that code simply runs as part of the component. However in most cases, especially in realtime components, the code must run in a specific sequence and at specific intervals. For example, inputs should be

---

<sup>1</sup>There has been some discussion about whether we really need all the integer types. Maybe they will be reduced or eliminated later. Most signals and pins will be either floats or bits.

read before calculations are performed on the input data, and outputs should not be written until the calculations are done. In these cases, the code is made available to the system in the form of one or more "functions". Each function is a block of code that performs a specific action. The system integrator can use "threads" to schedule a series of functions to be executed in a particular order and at specific time intervals.

**Thread:** A "thread" is a list of functions that runs at specific intervals as part of a realtime task. When a thread is first created, it has a specific time interval (period), but no functions. Functions can be added to the thread, and will be executed in order every time the thread runs.

For now a quick example will help get the concept across. We have a parport component named `hal_parport`. That component defines one or more HAL pins for each physical pin. The pins are described in that component's doc section - their names, how each pin relates to the physical pin, are they inverted, can you change polarity, etc. But that alone doesn't get the data from the HAL pins to the physical pins. It takes code to do that, and that is where functions come into the picture. The parport component needs at least two functions. One to read the physical input pins and update the HAL pins, the other to take data from the HAL pins and write it to the physical output pins. Both of these functions are part of the parport driver.

### 3.3 HAL components

Each HAL component is a piece of software with well defined inputs, outputs, and behaviour, that can be installed and interconnected as needed. This section lists available components and a brief description of what they do. Complete details for each component are available later in this document.

#### 3.3.1 External programs with HAL hooks

**motion** A realtime module that accepts NML motion commands and interacts with HAL

**iocontrol** A user space module that accepts NML I/O commands and interacts with HAL

**classicladder** A PLC using HAL for all I/O

**halui** A user space program that interacts with HAL and sends NML commands (note: right now experimental), it is intended to work as a full User Interface using external knobs & switches

#### 3.3.2 Internal Components

**stepgen** Software step pulse generator with position loop. See section ??

**freqgen** Software step pulse generator. See section ??

**encoder** Software based encoder counter. See section ??

**pid** Proportional/Integral/Derivative control loops. See section ??

**siggen** A sine/cosine/triangle/square wave generator for testing. See section ??

**supply** a simple source for testing

**blocks** assorted useful components (mux, demux, or, and, integ, ddt, limit, wcomp, etc.)

### 3.3.3 Hardware drivers

**hal\_ax5214h** A driver for the Axiom Measurement & Control AX5241H digital I/O board

**hal\_m5i20** Mesa Electronics 5i20 board

**hal\_motenc** Vital Systems MOTENC-100 board

**hal\_parport** PC parallel port. See section ??

**hal\_ppmc** Pico Systems family of controllers (PPMC, USC and UPC)

**hal\_stg** Servo To Go card (version 1 & 2)

**hal\_vti** Vigilant Technologies PCI ENCDAC-4 controller

### 3.3.4 Utilities

**halcmd** Command line tool for configuration and tuning. See section ??

**halgui** GUI tool for configuration and tuning (not implemented yet).

**halmeter** A handy multimeter for HAL signals. See section ??

**halscope** A full featured digital storage oscilloscope for HAL signals. See section ??

Each of these building blocks is described in detail in later chapters.

## 3.4 Tinkertoys, Erector Sets, Legos and the HAL

A first introduction to HAL concepts can be mind boggling. Building anything with blocks can be a challenge but some of the toys that we played with as kids can be an aid to building things with the HAL.

### 3.4.1 Tower

I'm watching as my son and his six year old daughter build a tower from a box full of random sized blocks, rods, jar lids and such. The aim is to see how tall they can make the tower. The narrower the base the more blocks left to stack on top. But the narrower the base, the less stable the tower. I see them studying both the next block and the shelf where they want to place it to see how it will balance out with the rest of the tower.

The notion of stacking cards to see how tall you can make a tower is a very old and honored way of spending spare time. At first read, the integrator may have gotten the impression that building a HAL was a bit like that. It can be but with proper planning an integrator can build a stable system as complex as the machine at hand requires.

### 3.4.2 Erector Sets<sup>2</sup>

What was great about the sets was the building blocks, metal struts and angles and plates, all with regularly spaced holes. You could design things and hold them together with the little screws and nuts.

---

<sup>2</sup>The Erector Set was an invention of AC Gilbert

I got my first erector set for my fourth birthday. I know the box suggested a much older age than I was. Perhaps my father was really giving himself a present. I had a hard time with the little screws and nuts. I really needed four arms, one each for the screwdriver, screw, parts to be bolted together, and nut. Perseverance, along with father's eventual boredom, got me to where I had built every project in the booklet. Soon I was lusting after the bigger sets that were also printed on that paper. Working with those regular sized pieces opened up a world of construction for me and soon I moved well beyond the illustrated projects.

Hal components are not all the same size and shape but they allow for grouping into larger units that will do useful work. In this sense they are like the parts of an Erector set. Some components are long and thin. They essentially connect high level commands to specific physical pins. Other components are more like the rectangular platforms upon which whole machines could be built. An integrator will quickly get beyond the brief examples and begin to bolt together components in ways that are unique to them.

### 3.4.3 Tinkertoys<sup>3</sup>

Wooden Tinker toys had a more humane feel than the cold steel of Erector Sets. The heart of construction with Tinker Toys was a round connector with eight holes equally spaced around the circumference. It also had a hole in the center that was perpendicular to all the holes around the hub.

Hubs were connected with rods of several different lengths. Builders would make large wheels by using these rods as spokes sticking out from the center hub.

My favorite project was a rotating space station. Short spokes radiated from all the holes in the center hub and connected with hubs on the ends of each spoke. These outer hubs were connected to each other with longer spokes. I'd spend hours dreaming of living in such a device, walking from hub to hub around the outside as it slowly rotated producing near gravity in weightless space. Supplies traveled through the spokes in elevators that transferred them to and from rockets docked at the center hub while they transferred their precious cargos.

The idea of one pin or component being the hub for many connections is also an easy concept within the HAL. Examples two and four (see section 4) connect the meter and scope to signals that are intended to go elsewhere. Less easy is the notion of a hub for several incoming signals but that is also possible with proper use of functions within that hub component that handle those signals as they arrive from other components.

Another thought that comes forward from this toy is a mechanical representation of HAL threads.

A thread might look a bit like a centipede, caterpillar, or earwig. A backbone of hubs, HAL components, strung together with rods, HAL signals. Each component takes in its own parameters and input pins and passes on output pins and parameters to the next component. Signals travel along the backbone from end to end and are added to or modified by each component in turn.

Threads are all about timing and doing a set of tasks from end to end. A mechanical representation is available with Tinkertoys also when we think of the length of the toy as a measure of the time taken to get from one end to the other. A very different thread or backbone is created by connecting the same set of hubs with different length rods. The total length of the backbone can be changed by the length of rods used to connect the hubs. The order of operations is the same but the time to get from beginning to end is very different.

### 3.4.4 A Lego Example<sup>4</sup>

When Lego blocks first arrived in our stores they were pretty much all the same size and shape. Sure there were half sized one and a few quarter sized as well but that rectangular one did most of

<sup>3</sup>Tinkertoy is now a registered trademark of the Hasbro company.

<sup>4</sup>The Lego name is a trademark of the Lego company.

the work. Lego blocks interconnected by snapping the holes in the underside of one onto the pins that stuck up on another. By overlapping layers, the joints between could be made very strong, even around corners or tees.

I watched my children and grandchildren build with legos – the same legos. There are a few thousand of them in an old ratty but heavy duty cardboard box that sits in a corner of the recreation room. It stays there in the open because it was too much trouble to put the box away and then get it back out for every visit and it is always used during a visit. There must be Lego parts in there from a couple dozen different sets. The little booklets that came with them are long gone but the magic of building with interlocking pieces all the same size is something to watch.

Notice the following description of building a set of motion components in the HAL and how much like a wall of lego blocks it is.

The motion module exports a pin for each axis in cartesean space, and another pin for each axis in joint space. When it is loaded, it automatically creates a "jumper" signal for each axis, and automatically connects those signals from the joint pin to the cartesean pin. So you automatically have "trivkins" as soon as you load the motion module. (trivkins – trivial kinematics is the case where each motor moves a single axis at 90 degrees to the others)

The motion module is like a pair of legos in a line end to end. Trivkins is just like a single block overlapping the two. The in and out motion pins are plugged into each other by the block resting above. But the parallel goes on.

If you need some other kinematics, you then load a specific kins component. This component "knows" the names of the pins that the motion module uses for each axis, both joint and cartesean. When the module loads, it again automatically creates signals and connects its own pins to the motion module's pins (which will disconnect the "jumpers"). It could also know the thread names used by the motion module, and could automatically add it's own functions to those threads.

Trivkins is removed so that the motion blocks can be spread apart and by using other blocks, a different bridge is built between input and output pins. In Lego terms, trivkins might be a gray block and xxkins might be a yellow block.

So the net result is that 24 HAL signals and two HAL functions are configured, with no action needed by the integrator other than loading the module. (24 signals are from 6 axis \* 2 because we have joint and cartesean \* 2 because we have forward and inverse kinematics. Two functions because we have forward and inverse.) Because these HAL signals exist, they can be metered or scoped or whatever for testing. But because both modules know their names and know how to automatically connect them, the integrator doesn't have to know or care.

This kind of automatic HAL configuration is possible because all kinematics modules "plug in" the same way.

### 3.5 Timing Issues In HAL

Threads is going to take a major intellectual push because unlike the physical wiring models between black boxes that we have said that HAL is based upon, simply connecting two pins with a hal-signal falls far short of the action of the physical case.

True relay logic consists of relays connected together, and when a contact opens or closes, current flows (or stops) immediately. Other coils may change state, etc, and it all just "happens". But in



PLC style ladder logic, it doesn't work that way. Usually in a single pass through the ladder, each rung is evaluated in the order in which it appears, and only once per pass. A perfect example is a single rung ladder, with a NC contact in series with a coil. The contact and coil belong to the same relay.

If this were a conventional relay, as soon as the coil is energized, the contacts begin to open and de-energize it. That means the contacts close again, etc, etc. The relay becomes a buzzer.

With a PLC, if the coil is OFF and the contact is closed when the PLC begins to evaluate the rung, then when it finishes that pass, the coil is ON. The fact that turning on the coil opens the contact feeding it is ignored until the next pass. On the next pass, the PLC sees that the contact is open, and de-energizes the coil. So the relay still switches rapidly between on and off, but at a rate determined by how often the PLC evaluates the rung.

In HAL, the function is the code that evaluates the rung(s). In fact a HAL-aware realtime version of ClassicLadder would export a function to do exactly that. Meanwhile, a thread is the thing that runs the function at specific time intervals. Just like you can choose to have a PLC evaluate all its rungs every 10mS, or every second, you can define HAL threads with different periods.

What distinguishes one thread from another is `_not_` what the thread does - that is determined by which functions are connected to it. The real distinction is simply how often a thread runs.

In EMC we might have a 15uS thread, a 1mS thread, and a 10mS thread. These would be created based on "Period", "ServoPeriod", and "TrajPeriod" respectively - the actual times would depend on the ini. That is one part of the config process, and although it could be done manually, it would normally be automatic.

The next step is to decide what each thread needs to do. Some of those decisions would also be automatic - the motion module would automatically connect its "PlanTrajectory" function to the TrajPeriod thread, and its "ControlMotion" function to the ServoPeriod thread.

Other connections would be made by the integrator (at least the first time). These might include hooking the STG driver's encoder read and DAC write functions to the servo thread, or hooking stepgen's function to the fast thread, along with the parport function(s) to write the steps to the port.

## 3.6 Dynamic Linking and Configuration

It is indeed possible to configure HAL with a form of dynamic linking. But it is different than DLLs as used by Microsoft(tm) or shared libraries as used in Linux. Both DLLs and shared libraries essentially say "Here I am, I have this code you might want to use", where "you" is other modules. Then when those other modules or programs are loaded, they say "I need a function called 'X', is there one?" and if the answer is YES, they link to it.

With HAL, a component still says "Here I am, I have this code you might want to use", but "you" is the system integrator. The integrator gets to decide what functions are used and doesn't have to worry about another module needing "function X" and not finding it.

HAL can follow the normal DLL model as well. Although most components will simply export pins, functions, and parameters, and then wait for the integrator (or a saved file) to interconnect them, we can write modules that (attempt to) make connections when they are installed. One specific place where this would work well is kinematics as illustrated in the Lego section [3.4.4](#).

# Chapter 4

## HAL Tutorial

### 4.1 Before we start

Configuration moves from theory to device – HAL device that is. For those who have had just a bit of computer programming, this section is the “Hello World” of the HAL. As noted above `halcmd` can be used to create a working system. It is a command line or text file tool for configuration and tuning. The following examples illustrate its setup and operation.

#### 4.1.1 Notation

Command line examples are presented in **bold typewriter** font. Responses from the computer will be in `typewriter` font. Text inside square brackets `[like-this]` is optional. Text inside angle brackets `<like-this>` represents a field that can take on different values, and the adjacent paragraph will explain the appropriate values. Text items separated by a vertical bar means that one or the other, but not both, should be present. All command line examples assume that you are in the `emc2/` directory, and paths will be shown accordingly when needed.

#### 4.1.2 Root Privileges

In the beginning days of HAL there was quite often the need to run things with root privileges. Most of those things were related to the fact that HAL uses kernel modules to do much of its work, and because it also can access hardware directly. Knowing that it is usually safer to avoid doing day-to-day work as root, most of those things were reworked so that very limited root privileges are required. If you are running a version of EMC2 and HAL more recent than early 2006, you can pretty much ignore this section.

To get around the need for root, `emc2` uses a small program called `emc_module_helper`, which during the build process gets `setuid` status (thus has root privileges). This small software module takes care of the `insmod/rmmod` commands that are needed to insert/delete modules from the HAL. You can use this program directly, but it's lots easier to use the **`halcmd loadrt & unloadrt`** commands ??

Here is an example of what happens when you don't have root privileges:

```
emc2$ bin/hal_parport 0278
PARPORT: ERROR: could not get I/O permission
emc2$
```

As an alternative to logging in as root, you can use the `sudo` command or the `su -c` command. The `sudo` command is very convenient to use, and does not require you to know the root password.

However, it needs to be configured by someone who does know the root password. The configuration determines who may use `sudo`, and what commands they can use it for. We will not

discuss `sudo` configuration here, try `man sudo` and/or talk to your system administrator. If `sudo` is properly configured, here is what happens:

```
emc2$ sudo bin/hal_parport 0278
Password: <enter your password>
PARPORT: installed driver for 1 ports
emc2$
```

As an added convenience, `sudo` remembers your password for a short time, so if you enter another `sudo` command within the time limit (usually 5 minutes) you don't have to type your password again.

The `su -c` command does not require configuration, but does require you to know the root password, and to type it in for every command. You also must put quotes around the command you are trying to run:

```
emc2$ su -c "bin/hal_parport 0278"
Password: <enter root password>
PARPORT: installed driver for 1 ports
emc2$
```

To avoid cluttering up the examples, we will not show `sudo` or `su -c`. Instead, commands that require root privileges will be preceded by `#`, and other commands will be preceded by `$`.

```
emc2$ ls bin
emc2# bin/hal_parport 0278
```

### 4.1.3 The RTAPI environment

RTAPI stands for Real Time Application Programming Interface. Many HAL components work in realtime, and all HAL components store data in shared memory so realtime components can access it. Normal Linux does not support realtime programming or the type of shared memory that HAL needs. Fortunately there are realtime operating systems (RTOS's) that provide the necessary extensions to Linux. Unfortunately, each RTOS does things a little differently.

To address these differences, the EMC team came up with RTAPI, which provides a consistent way for programs to talk to the RTOS. If you are a programmer who wants to work on the internals of EMC, you may want to study `emc2/src/rtapi/rtapi.h` to understand the API. But if you are a normal person all you need to know about RTAPI is that it (and the RTOS) needs to be loaded into the memory of your computer before you do anything with HAL.

For this tutorial, we are going to assume that you have successfully compiled the `emc2/` source tree. In that case, all you need to do is load the required RTOS and RTAPI modules into memory.

Just run the following command (needs root privileges):

```
emc2# scripts/realtime start
```

With the realtime OS and RTAPI loaded, we can move into the first example.

## 4.2 A Simple Example

### 4.2.1 Loading a realtime component

For the first example, we will use a HAL component called `siggen`, which is a simple signal generator. A complete description of the `siggen` component can be found in section ?? of this document. It is a realtime component, implemented as a Linux kernel module and located in the directory `emc2/rtdlib/`. To load `siggen` use the `halcmd loadrt siggen` command:

```
emc2$ bin/halcmd loadrt siggen
emc2$
```

## 4.2.2 Examining the HAL

Now that the module is loaded, it is time to introduce `halcmd`, the command line tool used to configure the HAL. This tutorial will introduce some `halcmd` features, for a more complete description try `man halcmd`, or see the `halcmd` reference in section ?? of this document. The first `halcmd` feature is the `show` command. This command displays information about the current state of the HAL. To show all installed components:

```
emc2$ bin/halcmd show comp
Loaded HAL Components:
ID  Type  Name
02  User  halcmd21345
01  RT    siggen
emc2$
```

Since `halcmd` itself is a HAL component, it will always show up in the list<sup>1</sup>. The list also shows the `siggen` component that we installed in the previous step. The “RT” under “Type” indicates that `siggen` is a realtime component.

Next, let’s see what pins `siggen` makes available:

```
emc2$ bin/halcmd show pin
Component Pins:
Owner  Type  Dir  Value      Name
02     float -W    0.00000e+00 siggen.0.cosine
02     float -W    0.00000e+00 siggen.0.sawtooth
02     float -W    0.00000e+00 siggen.0.sine
02     float -W    0.00000e+00 siggen.0.square
02     float -W    0.00000e+00 siggen.0.triangle
emc2$
```

This command displays all of the pins in the HAL - a complex system could have dozens or hundreds of pins. But right now there are only five pins. All five of these pins are floating point, and all five carry data out of the `siggen` component. Since we have not yet executed the code contained within the component, all the pins have a value of zero.

The next step is to look at parameters:

```
emc2$ bin/halcmd show param
Parameters:
Owner  Type  Dir  Value      Name
02     float -W    1.00000e+00 siggen.0.amplitude
02     float -W    1.00000e+00 siggen.0.frequency
02     float -W    0.00000e+00 siggen.0.offset
02     s32   R-    0          siggen.0.update.time
02     s32   RW    0          siggen.0.update.tmax
emc2$
```

The `show param` command shows all the parameters in the HAL. Right now each parameter has the default value it was given when the component was loaded. Note the column labeled `Dir`. The parameters labeled `-W` are writeable ones that are never changed by the component itself, instead they are meant to be changed by the user to control the component. We will see how to do this later. Parameters labeled `R-` are read only parameters. They can be changed only by the component. Finally, parameter labeled `RW` are read-write parameters. That means that they are changed by the component, but can also be changed by the user. Note: the parameters

---

<sup>1</sup>The number after `halcmd` in the component list is the process ID. It is possible to run more than one copy of `halcmd` at the same time (in different windows for example), so the PID is added to the end of the name to make it unique.

`siggen.0.update.time` and `siggen.0.update.tmax` are for debugging purposes, and won't be covered in this section.

Most realtime components export one or more functions to actually run the realtime code they contain. Let's see what function(s) `siggen` exported:

```
emc2$ bin/halcmd show funct
Exported Functions:
Owner CodeAddr  Arg      FP  Users  Name
  02   C48E31C4 C48D2054 YES    0  siggen.0.update
emc2$
```

The `siggen` component exported a single function. It requires floating point. It is not currently linked to any threads, so "users" is zero<sup>2</sup>.

### 4.2.3 Making realtime code run

To actually run the code contained in the function `siggen.0.update`, we need a realtime thread. Eventually `halcmd` will have a `newthread` command that can be used to create a thread, but that requires some significant internal changes. For now, we have a component called `threads` that is used to create a new thread. Lets create a thread called `test-thread` with a period of 1mS (1000000nS):

```
emc2$ bin/halcmd loadrt threads name1=test-thread period1=1000000
```

Let's see if that worked:

```
emc2$ bin/halcmd show thread
Realtime Threads:
  Period  FP  Name      (Time, Max-Time)
    999849 YES  test-thread ( 0, 0 )
emc2$
```

It did. The period is not exactly 1000000nS because of hardware limitations, but we have a thread that runs at approximately the correct rate, and which can handle floating point functions. The next step is to connect the function to the thread:

```
emc2$ bin/halcmd addf siggen.0.update test-thread
emc2$
```

Up till now, we've been using `halcmd` only to look at the HAL. However, this time we used the `addf` (add function) command to actually change something in the HAL. We told `halcmd` to add the function `siggen.0.update` to the thread `test-thread`, and if we look at the thread list again, we see that it succeeded:

```
emc2$ bin/halcmd show thread
Realtime Threads:
  Period  FP  Name      (Time, Max-Time)
    999849 YES  test-thread ( 0, 0 )
                          1 siggen.0.update
emc2$
```

There is one more step needed before the `siggen` component starts generating signals. When the HAL is first started, the thread(s) are not actually running. This is to allow you to completely configure the system before the realtime code starts. Once you are happy with the configuration, you can start the realtime code like this:

---

<sup>2</sup>The `codeaddr` and `arg` fields were used in development, and should probably be removed from the `halcmd` listing.

```
emc2$ bin/halcmd start
emc2$
```

Now the signal generator is running. Let's look at it's output pins:

```
emc2$ bin/halcmd show pin
Component Pins:
Owner  Type  Dir   Value      Name
02     float -W    5.61498e-01 siggen.0.cosine
02     float -W   -6.89775e-01 siggen.0.sawtooth
02     float -W    8.27478e-01 siggen.0.sine
02     float -W   -1.00000e+00 siggen.0.square
02     float -W    3.79549e-01 siggen.0.triangle
emc2$ bin/halcmd show pin
Component Pins:
Owner  Type  Dir   Value      Name
02     float -W    9.23063e-01 siggen.0.cosine
02     float -W   -8.74322e-01 siggen.0.sawtooth
02     float -W    3.84649e-01 siggen.0.sine
02     float -W   -1.00000e+00 siggen.0.square
02     float -W    7.48645e-01 siggen.0.triangle
emc2$
```

We did two `show pin` commands in quick succession, and you can see that the outputs are no longer zero. The sine, cosine, sawtooth, and triangle outputs are changing constantly. The square output is also working, however it simply switches from +1.0 to -1.0 every cycle, and it happened to be at -1.0 for both commands.

#### 4.2.4 Changing parameters

The real power of HAL is that you can change things. For example, we can use the `setp` command to set the value of a parameter. Let's change the amplitude of the signal generator from 1.0 to 5.0:

```
emc2$ bin/halcmd setp siggen.0.amplitude 5
emc2$
```

Check the parameters and pins again:

```
emc2$ bin/halcmd show param
Parameters:
Owner  Type  Dir   Value      Name
02     float -W    5.00000e+00 siggen.0.amplitude
02     float -W    1.00000e+00 siggen.0.frequency
02     float -W    0.00000e+00 siggen.0.offset
emc2$ bin/halcmd show pin
Component Pins:
Owner  Type  Dir   Value      Name
02     float -W   -1.66602e+00 siggen.0.cosine
02     float -W    1.95935e+00 siggen.0.sawtooth
02     float -W   -4.71428e+00 siggen.0.sine
02     float -W    5.00000e+00 siggen.0.square
02     float -W   -1.08130e+00 siggen.0.triangle
emc2$ bin/halcmd show pin
Component Pins:
Owner  Type  Dir   Value      Name
02     float -W   -3.82623e+00 siggen.0.cosine
```

```

02    float -W  -1.11309e+00  siggen.0.sawtooth
02    float -W   3.21869e+00  siggen.0.sine
02    float -W  -5.00000e+00  siggen.0.square
02    float -W  -2.77382e+00  siggen.0.triangle
emc2$

```

Note that the value of parameter `siggen.0.amplitude` has changed to 5.000, and that the pins now have larger values. The square wave output now switches from +5.0 to -5.0, and we happened to catch it switching this time.

### 4.2.5 Saving the HAL configuration

Most of what we have done with `halcmd` so far has simply been viewing things with the `show` command. However two of the commands actually changed things. As we design more complex systems with HAL, we will use many commands to configure things just the way we want them. HAL has the memory of an elephant, and will retain that configuration until we shut it down. But what about next time? We don't want to manually enter a bunch of commands every time we want to use the system. We can save the configuration of the entire HAL with a single command:

```

emc2$ bin/halcmd save
# components
loadrt threads name1=test-thread period1=1000000
loadrt siggen
# signals
# links
# parameter values
setp siggen.0.amplitude 5.00000e+00
setp siggen.0.frequency 1.00000e+00
setp siggen.0.offset 0.00000e+00
# realtime thread/function links
addf siggen.0.update test-thread
emc2$

```

The output of the `save` command is a sequence of HAL commands. If you start with an “empty” HAL and run all these commands, you will get the configuration that existed when the `save` command was issued. To save these commands for later use, we simply redirect the output to a file:

```

emc2$ bin/halcmd save >saved.hal
emc2$

```

### 4.2.6 Restoring the HAL configuration

To restore the HAL configuration stored in `saved.hal`, we need to execute all of those HAL commands. To do that, we use `halcmd -f <filename>` which reads commands from a file:

```

emc2$ bin/halcmd -f saved.hal
emc2$

```

## 4.3 Looking at the HAL with halmeter

You can build very complex HAL systems without ever using a graphical interface. However there is something satisfying about seeing the result of your work. The first and simplest GUI tool for the

HAL is halmeter. It is a very simple program that is the HAL equivalent of the handy Fluke multimeter (or Simpson analog meter for the old timers).

We will use the siggen component again to check out halmeter. If you just finished the previous example, then siggen is already loaded. If not, we can load it just like we did before:

```
emc2$ scripts/realtime start
emc2$ loadrt siggen
emc2$ loadrt threads name1=test-thread period1=1000000
emc2$ bin/halcmd addf siggen.0.update test-thread
emc2$ bin/halcmd start
emc2$ bin/halcmd setp siggen.0.amplitude 5
emc2$
```

### 4.3.1 Starting halmeter

At this point we have the siggen component loaded and running. It's time to start halmeter. Since halmeter is a GUI app, X must be running. We can start halmeter in the background by following it's name with a '&':

```
emc2$ bin/halmeter &
[1] 22093
emc2$
```

Since we started halmeter in the background, Linux prints its process id [1] 22093 and immediately returns to the shell prompt. At the same time, a halcmd window opens on your screen, looking something like figure 4.1. Note that you don't have to run halmeter in the background. If you omit '&', it will start and behave exactly the same, but you won't get your shell prompt back until you exit from halmeter.



Figure 4.1: Halmeter at startup, nothing selected

### 4.3.2 Using halmeter

The meter in figure 4.1 isn't very useful, because it isn't displaying anything. To change that, click on the 'Select' button, which will open the probe selection dialog (figure 4.2).

This dialog has three tabs. The first tab displays all of the HAL pins in the system. The second one displays all the signals, and the third displays all the parameters. We would like to look at the pin `siggen.0.triangle` first, so click on it then click the 'OK' button. The probe selection dialog will close, and the meter looks something like figure 4.3.

You should see the value changing as siggen generates its triangle wave. Halmeter refreshes its display about 5 times per second.

If you want to quickly look at a number of pins, you can use the 'Accept' button in the source selection dialog. Click on 'Select' to open the dialog again. This time, click on another pin, like `siggen.0.cosine`, and then click 'Accept'. When you click 'Accept', the meter immediately begins to display the newly selected item, but the dialog does not close. Try displaying a parameter instead





Figure 4.2: Halmeter source selection dialog



Figure 4.3: Halmeter displaying the value of a pin

of a pin. Click on the 'Parameters' tab, then select a parameter and click 'Accept' again. You can very quickly move the "meter probes" from one item to the next with a couple of clicks.

To shut down halmeter, just click the exit button.

If you want to look at more than one pin, signal, or parameter at a time, you can just start more halmeters. The halmeter window was intentionally made very small so you could have a lot of them on the screen at once. <sup>3</sup>

---

<sup>3</sup>Halmeter is due for a rewrite. The rewrite will do a number of things to make it nicer. Scientific notation will go away - it is a pain to read. Some form of ranging (including autoranging) will be added to allow it to display a wide range of numbers without using scientific notation. An "analog bar graph" display will also be added to give a quick indication of trends. When the rewrite is done, these screenshots and the accompanying text will be revised to match the new version.

## 4.4 A slightly more complex example.

Up till now we have only loaded one HAL component. But the whole idea behind the HAL is to allow you to load and connect a number of simple components to make up a complex system. The next example will use two components.

Before we can begin building this new example, we want to start with a clean slate. If you just finished one of the previous examples, we need to remove the all components and reload the RTAPI and HAL libraries:

```
emc2$ bin/halcmd unloadrt all
emc2$ scripts/realtime restart
emc2$
```

### 4.4.1 Installing the components

Now we are going to load the step pulse generator component. For a detailed description of this component refer to section ?? . For now, we can skip the details, and just run the following commands:

```
emc2$ bin/halcmd loadrt freqgen step_type=0,0
emc2$ bin/halcmd loadrt siggen
emc2$ bin/halcmd loadrt threads name1=fast fp1=0 period1=50000 name2=slow period2=1000
emc2$
```

The first command loads two step generators, both configured to generate stepping type 0. The second command loads our old friend siggen, and the third one creates two threads, a fast one with a period of 50 micro-seconds and a slow one with a period of 1mS. The fast thread doesn't support floating point functions.

As before, we can use `halcmd show` to take a look at the HAL. This time we have a lot more pins and parameters than before:

```
emc2$ bin/halcmd show pin
Component Pins:
Owner  Type  Dir  Value      Name
03     float -W   0.00000e+00 siggen.0.cosine
03     float -W   0.00000e+00 siggen.0.sawtooth
03     float -W   0.00000e+00 siggen.0.sine
03     float -W   0.00000e+00 siggen.0.square
03     float -W   0.00000e+00 siggen.0.triangle
02     s32   -W   0          freqgen.0.counts
02     bit   -W   FALSE     freqgen.0.dir
02     float -W   0.00000e+00 freqgen.0.position
02     bit   -W   FALSE     freqgen.0.step
02     float R-  0.00000e+00 freqgen.0.velocity
02     s32   -W   0          freqgen.1.counts
02     bit   -W   FALSE     freqgen.1.dir
02     float -W   0.00000e+00 freqgen.1.position
02     bit   -W   FALSE     freqgen.1.step
02     float R-  0.00000e+00 freqgen.1.velocity
emc2$ bin/halcmd show param
Parameters:
Owner  Type  Dir  Value      Name
03     float -W   1.00000e+00 siggen.0.amplitude
03     float -W   1.00000e+00 siggen.0.frequency
03     float -W   0.00000e+00 siggen.0.offset
```

```

02    u8    -W    1    (01)    freqgen.0.dirhold
02    u8    -W    1    (01)    freqgen.0.dirsetup
02    float R-    0.00000e+00    freqgen.0.frequency
02    float -W    0.00000e+00    freqgen.0.maxaccel
02    float -W    1.00000e+15    freqgen.0.maxfreq
02    float -W    1.00000e+00    freqgen.0.position-scale
02    s32   R-    0
02    u8    -W    1    (01)    freqgen.0.steplen
02    u8    -W    1    (01)    freqgen.0.stepspace
02    float -W    1.00000e+00    freqgen.0.velocity-scale
02    u8    -W    1    (01)    freqgen.1.dirhold
02    u8    -W    1    (01)    freqgen.1.dirsetup
02    float R-    0.00000e+00    freqgen.1.frequency
02    float -W    0.00000e+00    freqgen.1.maxaccel
02    float -W    1.00000e+15    freqgen.1.maxfreq
02    float -W    1.00000e+00    freqgen.1.position-scale
02    s32   R-    0
02    u8    -W    1    (01)    freqgen.1.steplen
02    u8    -W    1    (01)    freqgen.1.stepspace
02    float -W    1.00000e+00    freqgen.1.velocity-scale
emc2$

```

#### 4.4.2 Connecting pins with signals

What we have is two step pulse generators, and a signal generator. Now it is time to create some HAL signals to connect the two components. We are going to pretend that the two step pulse generators are driving the X and Y axis of a machine. We want to move the table in circles. To do this, we will send a cosine signal to the X axis, and a sine signal to the Y axis. The siggen module creates the sine and cosine, but we need “wires” to connect the modules together. In the HAL, “wires” are called signals. We need to create two of them. We can call them anything we want, for this example they will be `X_vel` and `Y_vel`. To create them we use the `newsig` command. We also need to specify the type of data that will flow through these “wires”, in this case it is floating point:

```

emc2$ bin/halcmd newsig X_vel float
emc2$ bin/halcmd newsig Y_vel float
emc2$

```

To make sure that worked, we can look at all the signals:

```

emc2$ bin/halcmd show sig
Signals:
Type      Value      Name
float     0.00000e+00 X_vel
float     0.00000e+00 Y_vel
emc2$

```

The next step is to connect the signals to component pins. The signal `X_vel` is intended to run from the cosine output of the signal generator to the velocity input of the first step pulse generator. The first step is to connect the signal to the signal generator output. To connect a signal to a pin we use the `linksp` command.

```

emc2$ bin/halcmd linksp X_vel siggen.0.cosine
emc2$

```

To see the effect of the `linksp` command, we show the signals again:

```
emc2$ bin/halcmd show sig
Signals:
Type      Value      Name
float     0.00000e+00 X_vel
                                     <== siggen.0.cosine
float     0.00000e+00 Y_vel
emc2$
```

When a signal is connected to one or more pins, the show command lists the pins immediately following the signal name. The “arrow” shows the direction of data flow - in this case, data flows from pin `siggen.0.cosine` to signal `X_vel`. Now let’s connect the `X_vel` to the velocity input of a step pulse generator:

```
emc2$ bin/halcmd linksp X_vel freqgen.0.velocity
emc2$
```

We can also connect up the Y axis signal `Y_vel`. It is intended to run from the sine output of the signal generator to the input of the second step pulse generator:

```
emc2$ bin/halcmd linksp Y_vel siggen.0.sine
emc2$ bin/halcmd linksp Y_vel freqgen.1.velocity
emc2$
```

Now let’s take a final look at the signals and the pins connected to them:

```
emc2$ bin/halcmd show sig
Signals:
Type      Value      Name
float     0.00000e+00 X_vel
                                     <== siggen.0.cosine
                                     ==> freqgen.0.velocity
float     0.00000e+00 Y_vel
                                     <== siggen.0.sine
                                     ==> freqgen.1.velocity
emc2$
```

The show sig command makes it clear exactly how data flows through the HAL. For example, the `X_vel` signal comes from pin `siggen.0.cosine`, and goes to pin `freqgen.0.velocity`.

### 4.4.3 Setting up realtime execution - threads and functions

Thinking about data flowing through “wires” makes pins and signals fairly easy to understand. Threads and functions are a little more difficult. Functions contain the computer instructions that actually get things done. Thread are the method used to make those instructions run when they are needed. First let’s look at the functions available to us:

```
emc2$ bin/halcmd show funct
Exported Functions:
Owner CodeAddr Arg FP Users Name
03 D89051C4 D88F10FC YES 0 siggen.0.update
02 D8902868 D88F1054 YES 0 freqgen.capture_position
02 D8902498 D88F1054 NO 0 freqgen.make_pulses
02 D89026F0 D88F1054 YES 0 freqgen.update_freq
emc2$
```

In general, you will have to refer to the documentation for each component to see what its functions do. In this case, the function `siggen.0.update` is used to update the outputs of the signal generator. Every time it is executed, it calculates the values of the sine, cosine, triangle, and square outputs. To make smooth signals, it needs to run at specific intervals.

The other three functions are related to the step pulse generators. The first one, `freqgen.capture_position`, is used for position feedback. It captures the value of an internal counter that counts the step pulses as they are generated. Assuming no missed steps, this counter indicates the position of the motor.

The main function for the step pulse generator is `freqgen.make_pulses`. Every time `make_pulses` runs it decides if it is time to take a step, and if so sets the outputs accordingly. For smooth step pulses, it should run as frequently as possible. Because it needs to run so fast, `make_pulses` is highly optimized and performs only a few calculations. Unlike the others, it does not need floating point math.

The last function, `freqgen.update_freq`, is responsible for doing scaling and some other calculations that need to be performed only when the frequency command changes.

What this means for our example is that we want to run `siggen.0.update` at a moderate rate to calculate the sine and cosine values. Immediately after we run `siggen.0.update`, we want to run `freqgen.update_freq` to load the new values into the step pulse generator. Finally we need to run `freqgen.make_pulses` as fast as possible for smooth pulses. Because we don't use position feedback, we don't need to run `freqgen.capture_position` at all.

We run functions by adding them to threads. Each thread runs at a specific rate. Let's see what threads we have available:

```
emc2$ bin/halcmd show thread
Realtime Threads:
  Period  FP  Name      ( 0, 0 )
    1005720 YES  slow
    50286 NO   fast      ( 0, 0 )
emc2$
```

The two threads were created when we loaded `threads`. The first one, `slow`, runs every millisecond, and is capable of running floating point functions. We will use it for `siggen.0.update` and `freqgen.update_freq`. The second thread is `fast`, which runs every 50 microseconds, and does not support floating point. We will use it for `freqgen.make_pulses`. To connect the functions to the proper thread, we use the `addf` command. We specify the function first, followed by the thread:

```
emc2$ bin/halcmd addf siggen.0.update slow
emc2$ bin/halcmd addf freqgen.update_freq slow
emc2$ bin/halcmd addf freqgen.make_pulses fast
emc2$
```

After we give these commands, we can run the `show thread` command again to see what happened:

```
emc2$ bin/halcmd show thread
Realtime Threads:
  Period  FP  Name      (Time, Max-Time)
    1005720 YES  slow      ( 0, 0 )
                1 siggen.0.update
                2 freqgen.update-freq
    50286 NO   fast      ( 0, 0 )
                1 freqgen.make-pulses
emc2$
```

Now each thread is followed by the names of the functions, in the order in which the functions will run.

#### 4.4.4 Setting parameters

We are almost ready to start our HAL system. However we still need to adjust a few parameters. By default, the `siggen` component generates signals that swing from +1 to -1. For our example that is fine, we want the table speed to vary from +1 to -1 inches per second. However the scaling of the step pulse generator isn't quite right. By default, it generates an output frequency of 1 step per second with an input of 1.000. It is unlikely that one step per second will give us one inch per second of table movement. Let's assume instead that we have a 5 turn per inch leadscrew, connected to a 200 step per rev stepper with 10x microstepping. So it takes 2000 steps for one revolution of the screw, and 5 revolutions to travel one inch. that means the overall scaling is 10000 steps per inch. We need to multiply the velocity input to the step pulse generator by 10000 to get the proper output. That is exactly what the parameter `freqgen.n.velocity-scale` is for. In this case, both the X and Y axis have the same scaling, so we set the scaling parameters for both to 10000:

```
emc2$ bin/halcmd setp freqgen.0.velocity-scale 10000
emc2$ bin/halcmd setp freqgen.1.velocity-scale 10000
emc2$
```

This velocity scaling means that when the pin `freqgen.0.velocity` is 1.000, the step generator will generate 10000 pulses per second (10KHz). With the motor and leadscrew described above, that will result in the axis moving at exactly 1.000 inches per second. This illustrates a key HAL concept - things like scaling are done at the lowest possible level, in this case in the step pulse generator. The internal signal `X_vel` is the velocity of the table in inches per second, and other components such as `siggen` don't know (or care) about the scaling at all. If we changed the leadscrew, or motor, we would change only the scaling parameter of the step pulse generator.

#### 4.4.5 Run it!

We now have everything configured and are ready to start it up. Just like in the first example, we use the `start` command:

```
emc2$ bin/halcmd start
emc2$
```

Although nothing appears to happen, inside the computer the step pulse generator is cranking out step pulses, varying from 10KHz forward to 10KHz reverse and back again every second. Later in this tutorial we'll see how to bring those internal signals out to run motors in the real world, but first we want to look at them and see what is happening.

### 4.5 Taking a closer look with halscope.

The previous example generates some very interesting signals. But much of what happens is far too fast to see with halmeter. To take a closer look at what is going on inside the HAL, we want an oscilloscope. Fortunately HAL has one, called `halscope`.

#### 4.5.1 Starting Halscope

Halscope has two parts - a realtime part that is loaded as a kernel module, and a user part that supplies the GUI and display. Before starting the GUI you must load the realtime part:

```
emc2$ bin/halcmd loadrt scope_rt
emc2$
```

Once the realtime part is loaded, we can start the GUI. Like halmeter, you can follow it with `&` so it runs in the background and you get your shell prompt back immediately:

```
emc2$ bin/halscope &
[2] 3678
emc2$
```

The scope GUI window will open, immediately followed by a “Realtime function not linked” dialog that looks like figure 4.4<sup>4</sup>.

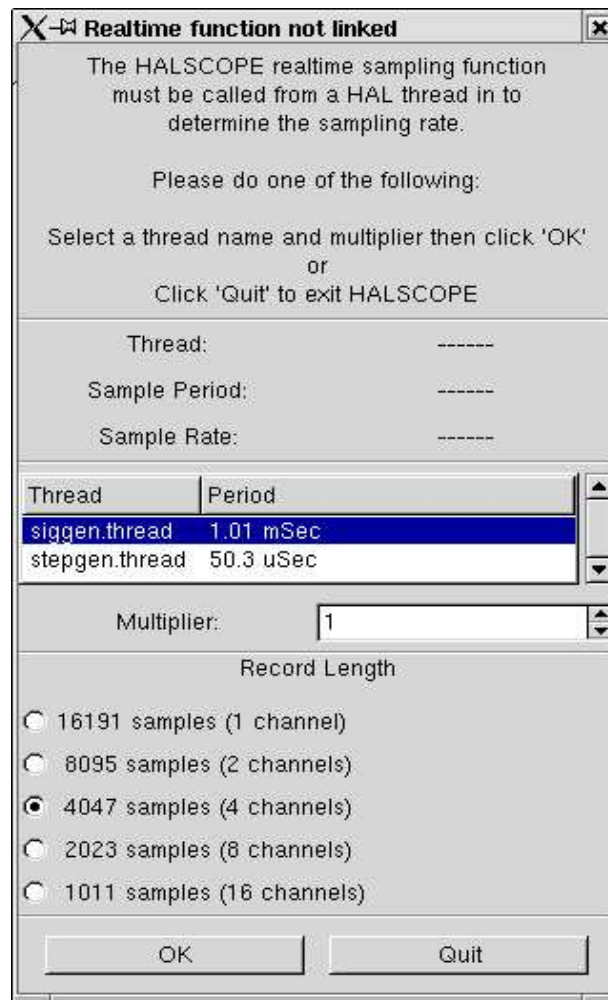


Figure 4.4: “Realtime function not linked” dialog

This dialog is where you set the sampling rate for the oscilloscope. For now we want to sample once per millisecond, so click on the 1.03mS thread “slow” (formerly “siggen.thread”, see footnote), and leave the multiplier at 1. We will also leave the record length at 4047 samples, so that we can use up to four channels at one time. When you select a thread and then click “OK”, the dialog disappears, and the scope window looks something like figure 4.5.

<sup>4</sup>Several of these screen captures refer to threads named “siggen.thread” and “stepgen.thread” instead of “slow” and “fast”. When the screenshots were captured, the “threads” component didn’t exist, and a different method was used to create threads, giving them different names. Also, the screenshots show pins, etc, as “stepgen.xxx” rather than “freqgen.xxx”. The original name of the freqgen module was stepgen, and I haven’t gotten around to re-doing all the screen shots since it was renamed. The name “stepgen” now refers to a different step pulse generator, one that accepts position instead of velocity commands. Both are described in detail later in this document.



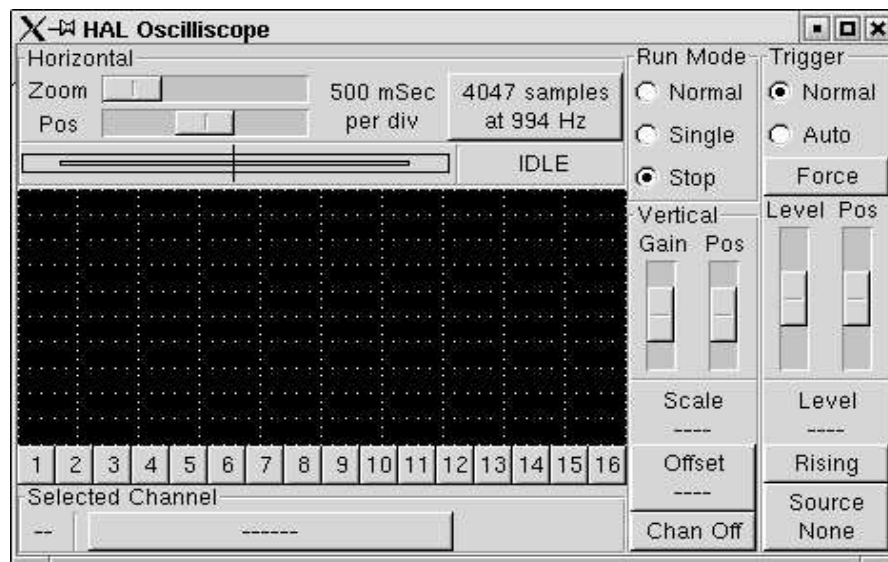


Figure 4.5: Initial scope window

### 4.5.2 Hooking up the “scope probes”

At this point, Halscope is ready to use. We have already selected a sample rate and record length, so the next step is to decide what to look at. This is equivalent to hooking “virtual scope probes” to the HAL. Halscope has 16 channels, but the number you can use at any one time depends on the record length - more channels means shorter records, since the memory available for the record is fixed at approximately 16,000 samples.

The channel buttons run across the bottom of the halscope screen. Click button “1”, and you will see the “Select Channel Source” dialog, figure 4.6. This dialog is very similar to the one used by Halmeter. We would like to look at the signals we defined earlier, so we click on the “Signals” tab, and the dialog displays all of the signals in the HAL (only two for this example).

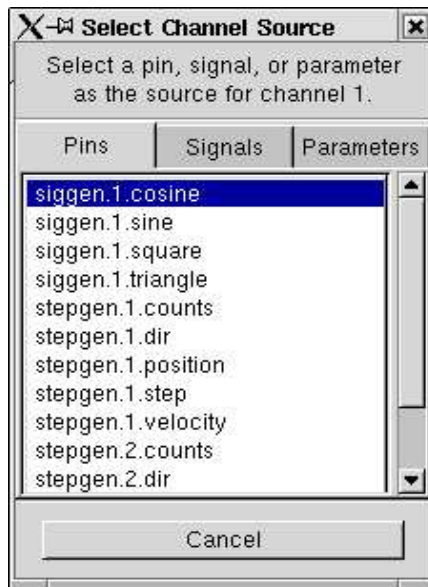


Figure 4.6: Select Channel Source dialog

To choose a signal, just click on it. In this case, we want to use channel 1 to display the signal “X\_vel”. When we click on “X\_vel”, the dialog closes and the channel is now selected. The channel 1 button is pressed in, and channel number 1 and the name “X\_vel” appear below the row of buttons. That display always indicates the selected channel - you can have many channels on the screen, but the selected one is highlighted, and the various controls like vertical position and scale always work on the selected one. To add a signal to channel 2, click the “2” button. When the dialog pops up, click the “Signals” tab, then click on “Y\_vel”.

We also want to look at the square and triangle wave outputs. There are no signals connected to those pins, so we use the “Pins” tab instead. For channel 3, select “siggen.0.triangle” and for channel 4, select “siggen.0.square”.

### 4.5.3 Capturing our first waveforms

Now that we have several probes hooked to the HAL, it's time to capture some waveforms. To start the scope, click the "Normal" button in the "Run Mode" section of the screen (upper right). Since we have a 4000 sample record length, and are acquiring 1000 samples per second, it will take halscope about 2 seconds to fill half of its buffer. During that time a progress bar just above the main screen will show the buffer filling. Once the buffer is half full, the scope waits for a trigger. Since we haven't configured one yet, it will wait forever. To manually trigger it, click the "Force" button in the "Trigger" section at the top right. You should see the remainder of the buffer fill, then the screen will display the captured waveforms. The result will look something like figure 4.7.

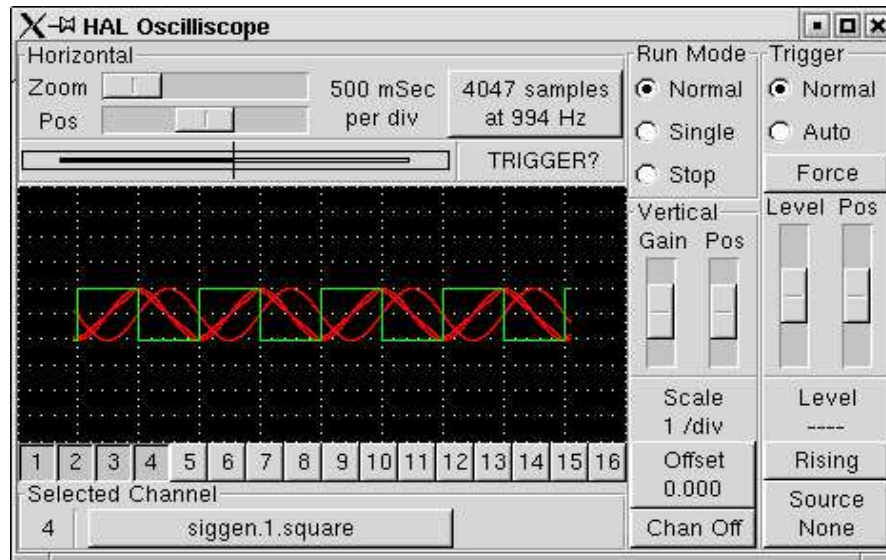


Figure 4.7: Captured Waveforms

The "Selected Channel" box at the bottom tells you that the green trace is the currently selected one, channel 4, which is displaying the value of the pin "siggen.1.square". Try clicking channel buttons 1 through 3 to highlight the other three traces.

### 4.5.4 Vertical Adjustments

The traces are rather hard to distinguish since all four are on top of each other. To fix this, we use the “Vertical” controls in the box to the right of the screen. These controls act on the currently selected channel. When adjusting the gain, notice that it covers a huge range - unlike a real scope, this one can display signals ranging from very tiny (pico-units) to very large (Tera-units). The position control moves the displayed trace up and down over the height of the screen only. For larger adjustments the offset button should be used (see the halscope reference in section ?? for details).

### 4.5.5 Triggering

Using the “Force” button is a rather unsatisfying way to trigger the scope. To set up real triggering, click on the “Source” button at the bottom right. It will pop up the “Trigger Source” dialog, which is simply a list of all the probes that are currently connected (Figure 4.8 ). Select a probe to use for triggering by clicking on it. For this example we will use channel 3, the triangle wave.

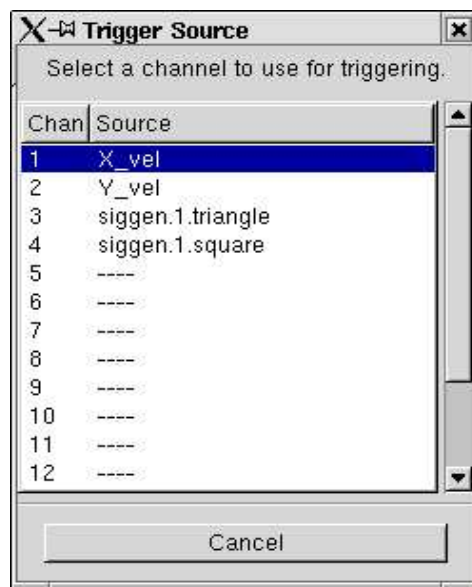


Figure 4.8: Trigger Source Dialog

After setting the trigger source, you can adjust the trigger level and trigger position using the sliders in the “Trigger” box along the right edge. The level can be adjusted from the top to the bottom of the screen, and is displayed below the sliders. The position is the location of the trigger point within the overall record. With the slider all the way down, the trigger point is at the end of the record, and halscope displays what happened before the trigger point. When the slider is all the way up, the trigger point is at the beginning of the record, displaying what happened after it was triggered. The trigger point is visible as a vertical line in the progress box above the screen. The trigger polarity can be changed by clicking the button just below the trigger level display. Note that changing the trigger position stops the scope, once the position is adjusted you restart the scope by clicking the “Normal” button in the “Run Mode” box.

Now that we have adjusted the vertical controls and triggering, the scope display looks something like figure 4.9.

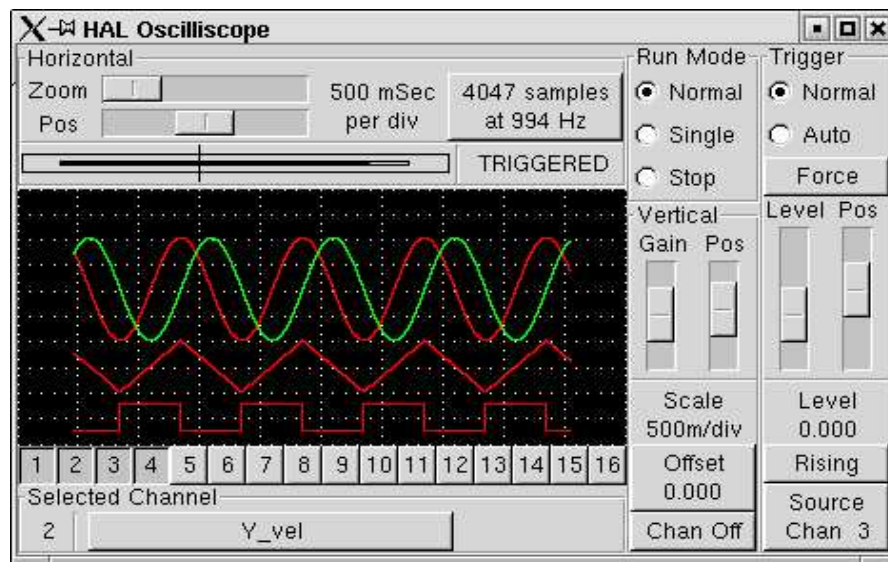


Figure 4.9: Waveforms with Triggering

### 4.5.6 Horizontal Adjustments

To look closely at part of a waveform, you can use the zoom slider at the top of the screen to expand the waveforms horizontally, and the position slider to determine which part of the zoomed waveform is visible. However, sometimes simply expanding the waveforms isn't enough and you need to increase the sampling rate. For example, we would like to look at the actual step pulses that are being generated in our example. Since the step pulses may be only 50uS long, sampling at 1KHz isn't fast enough. To change the sample rate, click on the button that displays the record length and sample rate to bring up the "Select Sample Rate" dialog, figure . For this example, we will click on the 50uS thread, "fast", which gives us a sample rate of about 20KHz. Now instead of displaying about 4 seconds worth of data, one record is 4000 samples at 20KHz, or about 0.20 seconds.

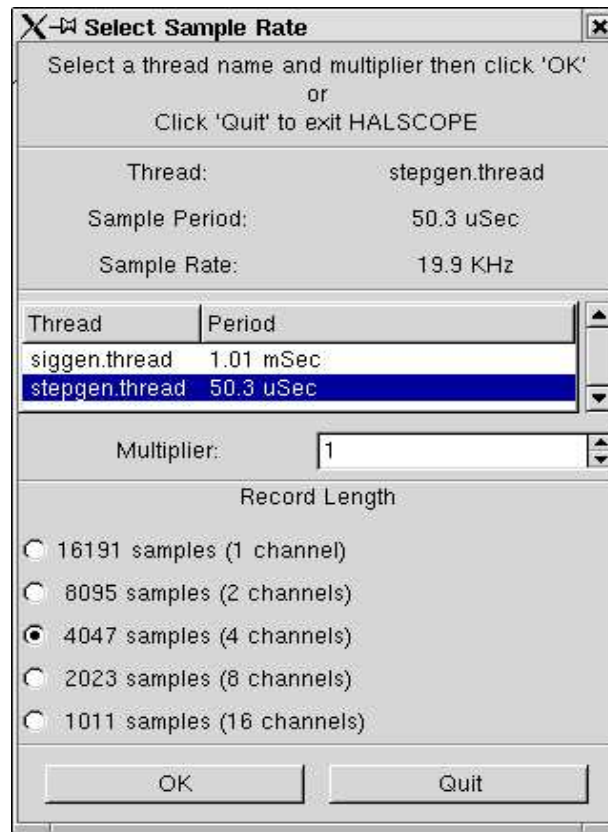


Figure 4.10: Sample Rate Dialog

### 4.5.7 More Channels

Now let's look at the step pulses. Halscope has 16 channels, but for this example we are using only 4 at a time. Before we select any more channels, we need to turn off a couple. Click on the channel 2 button, then click the "Off" button at the bottom of the "Vertical" box. Then click on channel 3, turn it off, and do the same for channel 4. Even though the channels are turned off, they still remember what they are connected to, and in fact we will continue to use channel 3 as the trigger source. To add new channels, select channel 5, and choose pin "stepgen.1.dir", then channel 6, and select "stepgen.1.step". Then click run mode "Normal" to start the scope, and adjust the horizontal zoom to 5mS per division. You should see the step pulses slow down as the velocity command (channel 1) approaches zero, then the direction pin changes state and the step pulses speed up again. You might want to increase the gain on channel 1 to about 20m per division to better see the change in the velocity command. The result should look like figure 4.11.

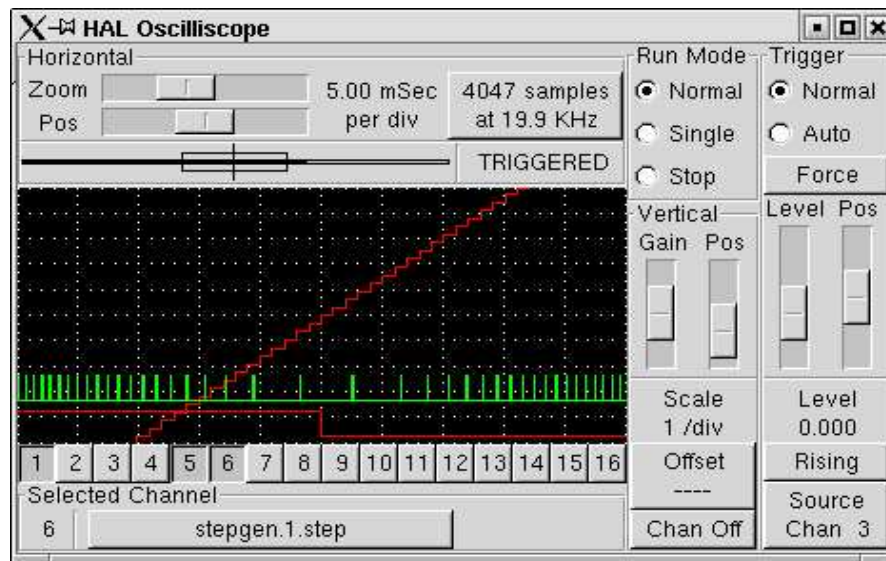


Figure 4.11: Looking at Step Pulses

**Part III**

**Using EMC2**





## **Chapter 5**

# **Using The TKEMC Graphical Interface**

# Chapter 6

## Using The MINI Graphical Interface

### 6.1 Introduction<sup>1</sup>

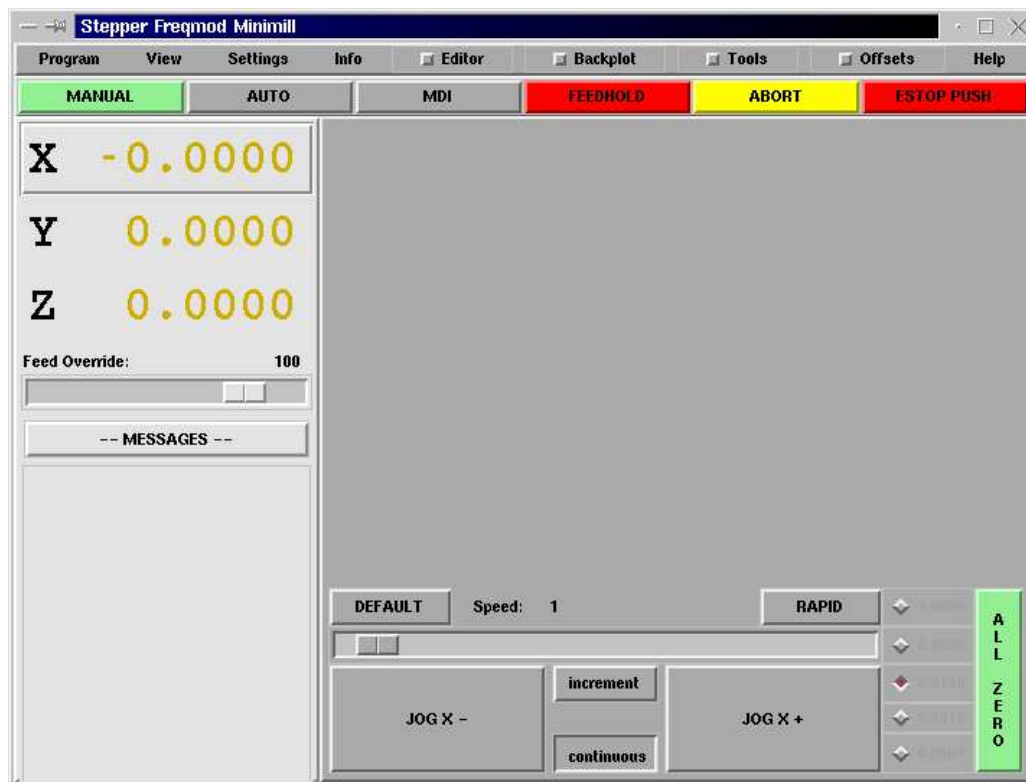


Figure 6.1: The Mini Graphical Interface

Mini was designed to be a full screen graphical interface. It was first written for the Sherline CNC but is available for anyone to use, copy, and distribute under the terms of the GPL copyright. Rather than popup new windows for each thing that an operator might want to do, Mini allows you to display these within the regular screen. Mini was written largely for the Sherline CNC mill. Parts of this chapter are copied from the instructions that were written for that mill by Joe Martin and Ray Henry.

<sup>1</sup>Much of this chapter quotes from a chapter of the Sherline CNC operators manual.

## 6.2 Screen layout

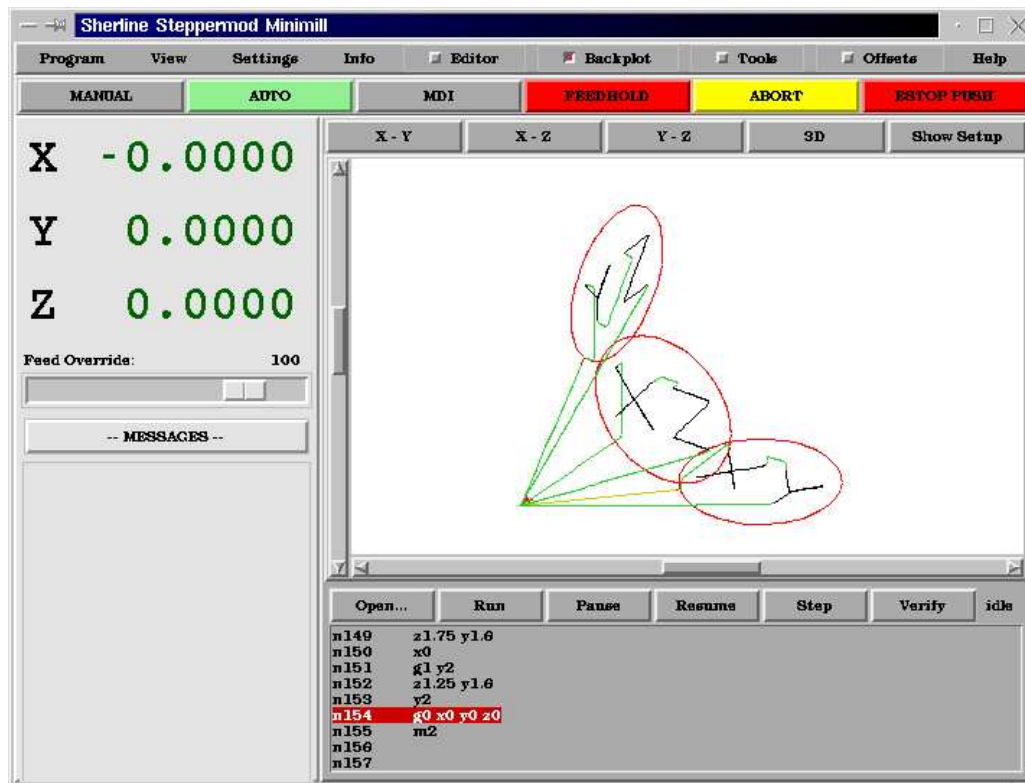


Figure 6.2: Mini Display for a Running EMC

The Mini screen is laid out in several sections. (See Figure 6.1 ) These include a menu across the top, a set of main control buttons just below the menu and two rather large columns of information that show the state of your machine and allow you to enter commands or programs. When you compare figure 6.1 with figure 6.2 you will see many differences. In the second figure

- each axis has been homed – the display numbers are dark green
- the EMC mode is auto – the auto button has a light green background
- the backplotter has been turned on – backplot is visible in the pop-in window
- the tool path from the program is showing in the display.

Once you start working with Mini you will quickly discover how easily it shows the conditions of the EMC and allows you to make changes to it.

## 6.3 Menu Bar

The first row is the menu bar across the top. Here you can configure the screen to display additional information. Some of the items in this menu are very different from what you may be accustomed to with other programs. You should take a few minutes and look under each menu item in order to familiarize yourself with the features that are there.

The menu includes each of the following sections and subsections.

**Program** This menu includes both reset and exit functions. Reset will return the EMC to the condition that it was in when it started. Some startup configuration items like the normal program units can be specified in the ini file.

**View** This menu includes several screen elements that can be added so that you can see additional information during a run. These include

**Position\_Type** This menu item adds a line above the main position displays that shows whether the displays are in inches or metric and whether they are Machine or Relative location and if they are Actual positions or Commanded positions. These can be changed using the Settings menu described below.

**Tool\_Info** This adds a line immediately below the main position displays that shows which tool has been selected and the length of offset applied.

**Offset\_Info** adds a line immediately below the tool info that shows what offsets have been applied. This is a total distance for each axis from machine zero.

**Show\_Restart** adds a block of buttons to the right of the program display in auto mode. These allow the operator to restart a program after an abort or estop. These will pop in whenever estop or abort is pressed but can be shown by the operator anytime auto mode is active by selecting this menu item.

**Hide\_Restart** removes the block of buttons that control the restart of a program that has been aborted or estopped.

**Show\_Split\_Right** changes the nature of the right hand column so that it shows both mode and pop-in information.

**Show\_Mode\_Full** changes the right hand column so that the mode buttons or displays fill the entire right side of the screen. In manual mode, running with mode full you will see spindle and lube control buttons as well as the motion buttons.

**Show\_Popin\_Full** changes the right hand column so that the popin fills the entire right side of the screen.

**Settings** These menu items allow the operator to control certain parameters during a run.

**Actual\_Position** sets the main position displays to actual(machine based) values.

**Commanded\_Position** sets the main position displays to the values that they were commanded to.

**Machine\_Position** sets the main position displays to the absolute distance from where the machine was homed.

**Relative\_Position** sets the main position displays to show the current position including any offsets like part zeros that are active. For more information on offsets see the chapter on coordinate systems.

**Info** lets you see a number of active things by writing their values into the MESSAGE pad.

**Program\_File** will write the currently active program file name.

**Editor\_File** will write the currently active file if the editor pop in is active and a file has been selected for editing.

**Parameter\_File** will write the name of the file being used for program parameters. You can find more on this in the chapters on offsets and using variables for programming.

**Tool\_File** will write the name of the tool file that is being used during this run.

**Active\_G-Codes** will write a list of all of the modal program codes that are active whenever this item is selected. For more information about modal codes see the introductory part programming chapter.

**Help** opens a text window pop in that displays the contents of the help file.

You will notice between the info menu and the help menu there are a set of four buttons. These are called check buttons because they have a small box that shows red if they have been selected. These four buttons, Editor, Backplot, Tools, and Offsets pop in each of these screens. If more than one pop-in is active (button shown as red) you can toggle between these pop-ins by right clicking your mouse.

## 6.4 Control Button Bar

Below the menu line is a horizontal line of control buttons. These are the primary control buttons for the interface. Using these buttons you can change mode from [MANUAL] to [AUTO] to [MDI] (Manual Data Input). These buttons show a light green background whenever that mode is active. You can also use the [FEEDHOLD], [ABORT], and [ESTOP] buttons to control a programmed move.

### 6.4.1 MANUAL

This button or pressing <F3> sets the EMC to Manual mode and displays an abbreviated set of buttons the operator can use to issue manual motion commands. The labels of the jog buttons change to match the active axis. Whenever Show\_Mode\_Full is active in in manual mode, you will see spindle and lube control buttons as well as the motion buttons. A keyboard <i> or <I> will switch from continuous jog to incremental jog. Pressing that key again will toggle the increment size through the available sizes.



Figure 6.3: Manual Mode Buttons

A button has been added to designate the present position as the home position. We felt that a machine of this type (Sherline 5400) would be simpler to operate if it didn't use a machine home position. This button will zero out any offsets and will home all axes right where they are.

Axis focus is important here. Notice (in figure 6.1) that in manual mode you see a line or *groove* around the X axis to highlight its position display. This groove says that X is the active axis. It will be the target for jog moves made with the *plus* and *minus* jog buttons. You can change axis focus by clicking on any other axis display. You can also change axis focus in manual mode if you press its name key on your keyboard. Case is not important here. [Y] or [y] will shift the focus to the Y axis. [A] or [a] will shift the focus to the A axis. To help you remember which axis will jog when you press the jog buttons, the active axis name is displayed on them.

The EMC can jog (move a particular axis) as long as you hold the button down when it is set for *continuous*, or it can jog for a preset distance when it is set for *incremental*. You can also jog the active axis by pressing the plus [+] or minus [-] keys on the keyboard. Again, case is not important for keyboard jogs. The two small buttons between the large jog buttons let you set which kind of jog you want. When you are in incremental mode, the distance buttons come alive. You can set a distance by pressing it with the mouse. You can toggle between distances by pressing [i] or [I] on the keyboard. Incremental jog

has an interesting and often unexpected effect. If you press the jog button while a jog is in progress, it will add the distance to the position it was at when the second jog command was issued. Two one-inch jog presses in close succession will not get you two inches of movement. You have to wait until the first one is complete before jogging again.

Jog speed is displayed above the slider. It can be set using the slider by clicking in the slider's open slot on the side you want it to move toward, or by clicking on the [Default] or [Rapid] buttons. This setting only affects the jog move while in manual mode. Once a jog move is initiated, jog speed has no effect on the jog. As an example of this, say you set jog mode to *incremental* and the increment to 1 inch. Once you press the [Jog] button it will travel that inch at the rate at which it started.

## 6.4.2 AUTO

When the Auto button is pressed, or <F4> on the keyboard, the EMC is changed into that mode, a set of the traditional auto operation buttons is displayed, and a small text window opens to show a part program. During run the active line will be displayed as white lettering on a red background.

In the auto mode, many of the keyboard keys are bound to controls. For example the numbers above the query keys are bound to feedrate override. The 0 sets 100%, 9 sets 90% and such.

Other keys work much the same as they do with the tkemc graphical interface.

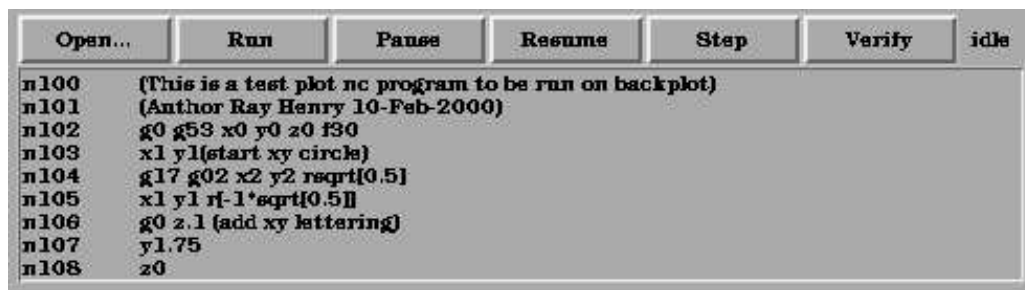


Figure 6.4: Auto Mode

Auto mode does not normally display the active or modal codes. If the operator wishes to check these, use menu Info -> Active\_G-Codes. This will write all modal codes onto the message scratch pad.

If abort or estop is pressed during a run a set of buttons displays to the right of the text that allows the operator to shift the restart line forward or backwards. If the restart line is not the last active line, it will be highlighted as white letters on a blue background. Caution, a very slow feedrate, and a finger poised over the pause button is advised during any program restart.

The real heart of CNC machine tool work is the auto mode. Sherline's auto mode displays the typical functions that people have come to expect from the EMC. Along the top are a set of buttons which control what is happening in auto mode. Below them is the window that shows the part of the program currently being executed. As the program runs, the active line shows in white letters on a red background. The first three buttons, [Open], [Run], and [Pause] do about what you'd expect. [Pause] will stop the run right where it is. The next button, [Resume], will restart motion. They are like feedhold if used this way. Once [Pause] is pressed and motion has stopped, [Step] will resume motion and continue it to the end of the current block. Press [Step] again to get the motion of the next block. Press [Resume] and the interpreter goes back to reading ahead and running the program. The combination of [Pause] and [Step] work a lot like single block mode on many controllers. The difference is that [Pause] does not let motion continue to the end of the current block. Feedrate Override ... can be very handy as you approach a first cut. Move in quickly at 100 percent, throttle back to 10% and toggle between [Feedhold] and

10% using the pause button. When you are satisfied that you've got it right, hit the zero to the right of nine and go.

The [Verify] button runs the interpreter through the code without initiating any motion. If Verify finds a problem it will stop the read near the problem block and put up some sort of message. Most of the time you will be able to figure out the problem with your program by reading the message and looking in the program window at the highlighted line. Some of the messages are not very helpful. Sometimes you will need to read a line or two ahead of the highlight to see the problem. Occasionally the message will refer to something well ahead of the highlight line. This often happens if you forget to end your program with an acceptable code like %, m2, m30, or m60.

### 6.4.3 MDI

The MDI button or <F5> sets the Manual Data Input mode. This mode displays a single line of text for block entry and shows the currently active modal codes for the interpreter.

MDI mode allows you to enter single blocks and have the interpreter execute them as if they were part of a program (kind of like a one-line program). You can execute circles, arcs, lines and such. You can even test sets of program lines by entering one block, waiting for that motion to end, and then enter the next block. Below the entry window, there is a listing of all of the current modal codes. This listing can be very handy. I often forget to enter a g00 before I command a motion. If nothing happens I look down there to see if g80 is in effect. G80 stops any motion. If it's there I remember to issue a block like g00 x0 y0 z0. In MDI you are entering text from the keyboard so none of the main keys work for commands to the running machine. [F1] will Estop the control.

Since many of the keyboard keys are needed for entry, most of the bindings that were available in auto mode are not available here.

### 6.4.4 [FEEDHOLD] – [CONTINUE]

Feedhold is a toggle. When the EMC is ready to handle or is handling a motion command this button shows the feedhold label on a red background. If feedhold has been pressed then it will show the continue label. Using it to pause motion has the advantage of being able to restart the program from where you stopped it. Feedhold will toggle between zero speed and whatever feedrate override was active before it was pressed. This button and the function that it activates is also bound to the pause button on most keyboards.

### 6.4.5 [ABORT]

The abort button stops any motion when it is pressed. It also removes the motion command from the EMC. No further motions are cued up after this button is pressed. If you are in auto mode, this button removes the rest of the program from the motion cue. It also records the number of the line that was executing when it was pressed. You can use this line number to restart the program after you have cleared up the reasons for pressing it.

### 6.4.6 [ESTOP]

The estop button is also a toggle but it works in three possible settings.

- When Mini starts up it will show a raised button with red background with black letters that say "ESTOP PUSH." This is the correct state of the machine when you want to run a program or jog an axis. Estop is ready to work for you when it looks like this.



- If you push the estop button while a motion is being executed, you will see a recessed gray button that says “ESTOPPED.” You will not be able to move an axis or do any work from the Mini gui when the estop button displays this way. Pressing it with your mouse will return Mini to normal ready condition.
- A third view is possible here. A recessed green button means that estop has been take off but the machine has not been turned on. Normally this only happens when <F1> estop has been pressed but <F2> has not been pressed.

Joe Martin says, “When all else fails press a software [ESTOP].” This does everything that abort does but adds in a reset so that the EMC returns to the standard settings that it wakes up on. If you have an external estop circuit that watches the relevant parallel port or DIO pin, a software estop can turn off power to the motors.

Most of the time, when we abort or EStop it’s because something went wrong. Perhaps we broke a tool and want to change it. We switch to manual mode and raise the spindle, change tools, and assuming that we got the length the same, get ready to go on. If we return the tool to the same place where the abort was issued, the EMC will work perfectly. It is possible to move the restart line back or ahead of where the abort happened. If you press the [Back] or [Ahead] buttons you will see a blue highlight that shows the relationship between the abort line and the one on which the EMC will start up again. By thinking through what is happening at the time of the restart you can place the tool tip where it will resume work in an acceptable manner. You will need to think through things like tool offsets barriers to motion along a diagonal line and such before you press the [Restart] button.

## 6.5 Left Column

There are two columns below the control line. The left side of the screen displays information of interest to the operator. There are very few buttons to press here.

### 6.5.1 Axis Position Displays

The axis position displays work exactly like they do with tkemc. The color of the letters is important.

- Red indicates that the machine is sitting on a limit switch or the polarity of a min or max limit is set wrong in the ini file.
- Yellow indicates that the machine is ready to be homed.
- Green indicates that the machine has been homed.

The position can be changed to display any one of several values by using the menu settings. The startup or default settings can be changed in the ini file so these displays wake up just the way that you want them.

### 6.5.2 Feedrate Override

Immediately below the axis position displays is the feedrate override slider. You can operate feed rate override and feedhold in any mode of operation. Override will change the speed of jogs or feed rate in manual or MDI modes. You can adjust feed rate override by grabbing the slider with your mouse and dragging it along the groove. You can also change feed rate a percent at a time by clicking in the slider’s groove. In auto mode you can also set feed override in 10% increments by pressing the top row of numbers. This slider is a handy visual reference to how much override is being applied to programmed feedrate.

### 6.5.3 Messages

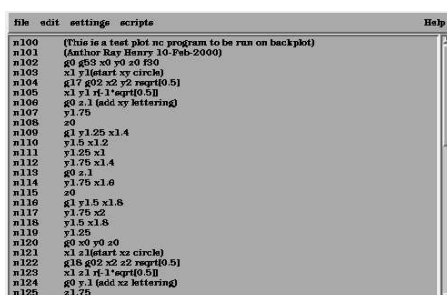
The message display located under the axis positions is a sort of scratch pad for the EMC. If there are problems it will report them there. If you try to home or move an axis when the [ESTOP] button is pressed, you'll get a message that says something about commanding motion when the EMC is not ready. If an axis faults out for something like falling behind, the message pad will show what happened. If you want to remind an operator to change a tool, for example, you can add a line of code to your program that will display in the message box. An example might be (`msg, change to tool #3 and press resume`). This line of code, included in a program, will display "change to tool #3 and press resume" in the message box. The word `msg`, (with comma included) is the command to make this happen; without `msg`, the message wouldn't be displayed. It will still show in the auto modes' display of the program file.

To erase messages simply click the message button at the top of the pad or on the keyboard hold down the [Alt] key and press the [m] key.

## 6.6 Right Column

The right column is a general purpose place to display and work. Here you can see the modal buttons and text entry or displays. Here you can view a plot of the tool path that will be commanded by your program. You can also write programs and control tools and offsets here. The modal screens have been described above. Each of the popin displays are described in detail below.

### 6.6.1 Program Editor



```

file  edit  settings  scripts  Help
n100 (This is a test plot nc program to be run on backplot)
n101 (Author Ray Henry 10-Feb-2000)
n102 g0 g53 z0 z0 z0 z0
n103 x1 y1(start xy circle)
n104 g17 g02 z2 z2 wpr(0.5)
n105 x1 z1 r:1*sqrt(0.5)
n106 g0 z:1 (add xy lettering)
n107 y1.75
n108 z0
n109 g1 y1.25 x1.4
n110 y1.5 x1.2
n111 y1.25 x1
n112 y1.75 x1.4
n113 g0 z:1
n114 y1.75 x1.6
n115 z0
n116 g1 y1.5 x1.8
n117 y1.75 x2
n118 y1.5 x1.8
n119 y1.25
n120 g0 x0 y0 z0
n121 x1 z:(start xx circle)
n122 g18 g02 z2 z2 wpr(0.5)
n123 x1 z1 r:1*sqrt(0.5)
n124 g0 z:1 (add xx lettering)
n125 z1.75

```

Figure 6.5: Mini Text Editor

The editor is rather limited compared to many modern text editors. It does not have *undo* nor *paste* between windows with the clipboard. These were eliminated because of interaction with a running program. Future releases will replace these functions so that it will work the way you've come to expect from a text editor. It is included because it has the rather nice feature of being able to number and renumber lines in the way that the interpreter expects of a file. It will also allow you to cut and paste from one part of a file to another. In addition, it will allow you to save your changes and submit them to the EMC interpreter with the same menu click. You can work on a file in here for a while and then save and load if the EMC is in Auto mode. If you have been running a file and find that you need to edit it, that file will be placed in the editor when you click on the editor button on the top menu.

### 6.6.2 Backplot Display

Backplot [Backplot] will show the tool path that can be viewed from a chosen direction. '3-D' is the default. Other choices and controls are displayed along the top and right side of the pop-in. If you are in the middle of a cut when you press one of these control buttons the machine will pause long enough to re-compute the view.

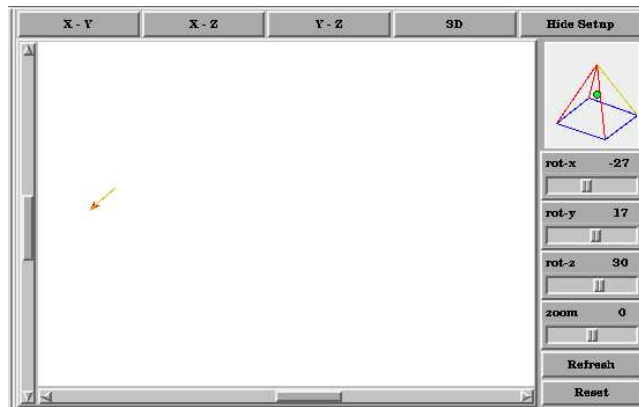


Figure 6.6: Mini's Backplotter

Along the right side of the pop-in there is a small pyramid shaped graphic that tries to show the angle you are viewing the tool path from. Below it are a series of sliders that allow you to change the angle of view and the size of the plot. You can rotate the little position angle display with these. They take effect when you press the [Refresh] button. The [Reset] button removes all of the paths from the display and readies it for a new run of the program but retains your settings for that session.

If backplot is started before a program is started, it will try to use some color lines to indicate the kind of motion that was used to make it. A green line is a rapid move. A black line is a feedrate move. Blue and red indicate arcs in counterclockwise and clockwise directions.

The backplotter with Mini allows you to zoom and rotate views after you have run your program but it is not intended to store a tool path for a long period of time.

### 6.6.3 Tool Page

The tool page is pretty much like the others. You can set length and diameter values here and they become effective when you press the [Enter] key. You will need to set up your tool information before you begin to run a program. You can't change tool offsets while the program is running or when the program is paused.

TOOL SETUP			
Click or tab to edit. Press enter to return to keyboard machine control.			
TOOL NUMBER	LENGTH	DIAMETER	COMMENT
1	1.456	0.250	Drill
2	1.000	0.4968	End Mill
3	0.0	0.0	empty
4	0.0	0.0	empty
5	0.0	0.0	empty
6	0.0	0.0	empty

Figure 6.7: Mini Tool Display

The [Add Tools] and [Remove Tools] buttons work on the bottom of the tool list so you will want to fill in tool information in descending order. Once a new tool has been added, you can use it in a program with the usual G-code commands. There is a 32 tool limit in the current EMC configuration files but you will run out of display space in Mini long before you get there. (Hint You can use menu -> view -> show popin full to see more tools if you need.)

### 6.6.4 Offset Page

The offset page can be used to display and setup work offsets. The coordinate system is selected along the left hand side of the window. Once you have selected a coordinate system you can enter values or move an axis to a teach position. You can also teach using an edfinder by adding the

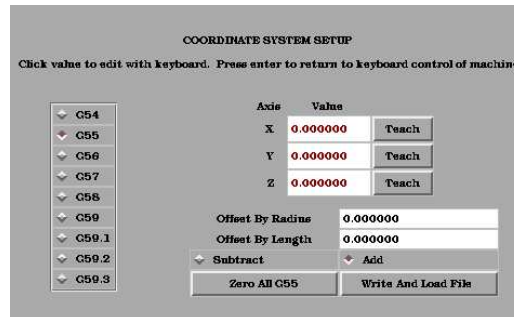


Figure 6.8: Mini Offset Display

radius and length to the offset\_by widgets. When you do this you may need to add or subtract the radius depending upon which surface you choose to touch from. This is selected with the add or subtract radiobuttons below the offset windows.

The zero all for the active coordinate system button will remove any offsets that you have showing but they are not set to zero in the variable file until you press the write and load file button as well. This write and load file button is the one to use when you have set all of the axis values that you want for a coordinate system.

## 6.7 Keyboard Bindings

A number of the bindings used with tkemc have been preserved with mini. A few of the bindings have been changed to extend that set or to ease the operation of a machine using this interface. Some keys operate the same regardless of the mode. Others change with the mode that EMC is operating in.

### 6.7.1 Common Keys

**Pause** Toggle feedhold

**Escape** abort motion

**F1** toggle estop/estop reset state

**F2** toggle machine off/machine on state

**F3** manual mode

**F4** auto mode

**F5** MDI mode

**F6** reset interpreter

The following only work for machines using auxiliary I/O

**F7** toggle mist on/mist off

**F8** toggle flood on/flood off

**F9** toggle spindle forward/off

**F10** toggle spindle reverse/off

**F11** decrease spindle speed

**F12** increase spindle speed

### 6.7.2 Manual Mode

**1-9 0** set feed override to 10%-90%, 0 is 100%

~ set feed override to 0 or feedhold

**x** select X axis

**y** select Y axis

**z** select Z axis

**a** select A axis

**b** select B axis

**c** select C axis

**Left Right Arrow** jog X axis

**Up Down Arrow** jog Y axis

**Page Up Down** jog Z axis

- \_ jog the active axis in the minus direction

+ = jog the active axis in the plus direction.

**Home** home selected axis

**i I** toggle through jog increments

The following only work with a machine using auxiliary I/O

**b** take spindle brake off

**Alt-b** put spindle brake on

### 6.7.3 Auto Mode

**1-9,0** set feed override to 10%-90%, 0 is 100%

~ set feed override to 0 or feedhold

**o/O** open a program

**r/R** run an opened program

**p/P** pause an executing program

**s/S** resume a paused program

**a/A** step one line in a paused program

## 6.8 Misc

One of the features of Mini is that it displays any axis above number 2 as a rotary and will display degree units for it. It also converts to degree units for incremental jogs when a rotary axis has the focus.

# Chapter 7

## Machining Center Overview

This section gives a brief description of how a machining center is viewed from the input and output ends of the Interpreter. It is assumed the reader is already familiar with machining centers.

Both the RS274/NGC input language and the output canonical machining functions have a view of (1) mechanical components of a machining center being controlled and (2) what activities of the machining center may be controlled, and what data is used in control.

The view here includes some items that a given machining center may not have, such as a pallet shuttle. The RS274/NGC language and canonical machining functions may be used with such a machine provided that no NC program used with the controller includes commands intended to activate physical capabilities the machine does not have. For such a machine, it would be useful to modify the Interpreter so it will reject input commands and will not produce output canonical function calls addressed to non-existent equipment.

### 7.1 Mechanical Components

A machining center has many mechanical components that may be controlled or may affect the way in which control is exercised. This section describes the subset of those components that interact with the Interpreter. Mechanical components that do not interact directly with the Interpreter, such as the jog buttons, are not described here, even if they affect control.

#### 7.1.1 Linear Axes

A machining center has independent mechanisms<sup>1</sup> for producing relative linear motion of the tool and workpiece in three mutually orthogonal directions. These are the X, Y and Z axes.

#### 7.1.2 Rotational axes

Three additional independent mechanisms produce relative rotation of the workpiece and the tool around an axis. These mechanisms (often a rotary table on which the workpiece is mounted or a drum on which the spindle is mounted) are called rotational axes and labelled A, B, and C. The A-axis is parallel to the X-axis. B is parallel to the Y-axis, and C parallel to the Z-axis<sup>2</sup>. Each rotational mechanism may or may not have a mechanical limit on how far it can rotate.

---

<sup>1</sup>If the motion of mechanical components is not independent, as with hexapod machines, the RS274/NGC language and the canonical machining functions will still be usable, as long as the lower levels of control know how to control the actual mechanisms to produce the same relative motion of tool and workpiece as would be produced by independent axes.

<sup>2</sup>The requirement of parallelism is not used by either language, so both languages are usable if any rotational axis is not parallel to any linear axis. Rotational axis commands flow through both languages to lower levels of control without significant change in nature.

### **7.1.3 Spindle**

A machining center has a spindle which holds one cutting tool, probe, or other item. The spindle can rotate in either direction, and it can be made to rotate at a constant rate, which may be changed. Except on machines where the spindle may be moved by moving a rotational axis, the axis of the spindle is kept parallel to the Z-axis and is coincident with the Z-axis when X and Y are zero. The spindle can be stopped in a fixed orientation or stopped without specifying orientation.

### **7.1.4 Coolant**

A machining center has components to provide mist coolant and/or flood coolant.

### **7.1.5 Pallet Shuttle**

A machining center has a pallet shuttle system. The system has two movable pallets on which workpieces can be fixtured. Only one pallet at a time is in position for machining.

### **7.1.6 Tool Carousel**

A machining center has a tool carousel with slots for tools fixed in tool holders.

### **7.1.7 Tool Changer**

A machining center has a mechanism for changing tools (fixed in tool holders) between the spindle and the tool carousel.

### **7.1.8 Message Display**

A machining center has a device that can display messages.

### **7.1.9 Feed and Speed Override Switches**

A machining center has separate feed and speed override switches, which let the operator specify that the actual feed rate or spindle speed used in machining should be some percentage of the programmed rate. See Section [7.3.1](#).

### **7.1.10 Block Delete Switch**

A machining center has a block delete switch. See Section [7.3.2](#).

### **7.1.11 Optional Program Stop Switch**

A machining center has an optional program stop switch. See Section [7.3.3](#).

## **7.2 Control and Data Components**

### **7.2.1 Linear Axes**

The X, Y, and Z axes form a standard right-handed coordinate system of orthogonal linear axes. Positions of the three linear motion mechanisms are expressed using coordinates on these axes.

## 7.2.2 Rotational Axes

The rotational axes are measured in degrees as wrapped linear axes in which the direction of positive rotation is counterclockwise when viewed from the positive end of the corresponding X, Y, or Z-axis. By “wrapped linear axis,” we mean one on which the angular position increases without limit (goes towards plus infinity) as the axis turns counterclockwise and decreases without limit (goes towards minus infinity) as the axis turns clockwise. Wrapped linear axes are used regardless of whether or not there is a mechanical limit on rotation.

Clockwise or counterclockwise is from the point of view of the workpiece. If the workpiece is fastened to a turntable which turns on a rotational axis, a counterclockwise turn from the point of view of the workpiece is accomplished by turning the turntable in a direction that (for most common machine configurations) looks clockwise from the point of view of someone standing next to the machine<sup>3</sup>

## 7.2.3 Controlled Point

The controlled point is the point whose position and rate of motion are controlled. When the tool length offset is zero (the default value), this is a point on the spindle axis (often called the gauge point) that is some fixed distance beyond the end of the spindle, usually near the end of a tool holder that fits into the spindle. The location of the controlled point can be moved out along the spindle axis by specifying some positive amount for the tool length offset. This amount is normally the length of the cutting tool in use, so that the controlled point is at the end of the cutting tool.

## 7.2.4 Coordinate Linear Motion

To drive a tool along a specified path, a machining center must often coordinate the motion of several axes. We use the term “coordinated linear motion” to describe the situation in which, nominally, each axis moves at constant speed and all axes move from their starting positions to their end positions at the same time. If only the X, Y, and Z axes (or any one or two of them) move, this produces motion in a straight line, hence the word “linear” in the term. In actual motions, it is often not possible to maintain constant speed because acceleration or deceleration is required at the beginning and/or end of the motion. It is feasible, however, to control the axes so that, at all times, each axis has completed the same fraction of its required motion as the other axes. This moves the tool along same path, and we also call this kind of motion coordinated linear motion.

Coordinated linear motion can be performed either at the prevailing feed rate, or at traverse rate. If physical limits on axis speed make the desired rate unobtainable, all axes are slowed to maintain the desired path.

## 7.2.5 Feed Rate

The rate at which the controlled point or the axes move is nominally a steady rate which may be set by the user. In the Interpreter, the interpretation of the feed rate is as follows unless inverse time feed rate mode is being used in the RS274/NGC view (see Section 9.19).

1. 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.
2. For motion of one rotational axis with X, Y, and Z axes not moving, the feed rate means degrees per minute rotation of the rotational axis.
3. For motion of two or three rotational axes with X, Y, and Z axes not moving, the rate is applied as follows. Let  $dA$ ,  $dB$ , and  $dC$  be the angles in degrees through which the A, B, and C axes, respectively, must move. Let  $D = \sqrt{(dA)^2 + (dB)^2 + (dC)^2}$ . Conceptually,  $D$  is a measure of

---

<sup>3</sup>If the parallelism requirement is violated, the system builder will have to say how to distinguish clockwise from counterclockwise.



total angular motion, using the usual Euclidean metric. Let  $T$  be the amount of time required to move through  $D$  degrees at the current feed rate in degrees per minute. The rotational axes should be moved in coordinated linear motion so that the elapsed time from the start to the end of the motion is  $T$  plus any time required for acceleration or deceleration.

### 7.2.6 Coolant

Flood coolant and mist coolant may each be turned on independently. The RS274/NGC language turns them off together (see Section 10.4).

### 7.2.7 Dwell

A machining center may be commanded to dwell (i.e., keep all axes unmoving) for a specific amount of time. The most common use of dwell is to break and clear chips, so the spindle is usually turning during a dwell. Regardless of the Path Control Mode (see Section 7.2.15) the machine will stop exactly at the end of the previous programmed move, as though it was in exact path mode.

### 7.2.8 Units

Units used for distances along the X, Y, and Z axes may be measured in millimeters or inches. Units for all other quantities involved in machine control cannot be changed. Different quantities use different specific units. Spindle speed is measured in revolutions per minute. The positions of rotational axes are measured in degrees. Feed rates are expressed in current length units per minute or in degrees per minute, as described in Section 7.2.5.

### 7.2.9 Current Position

The controlled point is always at some location called the “current position,” and the controller always knows where that is. The numbers representing the current position must be adjusted in the absence of any axis motion if any of several events take place:

1. Length units are changed.
2. Tool length offset is changed.
3. Coordinate system offsets are changed.

### 7.2.10 Selected Plane

There is always a “selected plane”, which must be the XY-plane, the YZ-plane, or the XZ-plane of the machining center. The Z-axis is, of course, perpendicular to the XY-plane, the X-axis to the YZ-plane, and the Y-axis to the XZ-plane.

### 7.2.11 Tool Carousel

Zero or one tool is assigned to each slot in the tool carousel.

### 7.2.12 Tool Change

A machining center may be commanded to change tools.

### 7.2.13 Pallet Shuttle

The two pallets may be exchanged by command.

### 7.2.14 Feed and Speed Override Switches

The feed and speed override switches may be enabled (so they work as expected) or disabled (so they have no effect on the feed rate or spindle speed). The RS274/NGC language has one command that enables both switches and one command that disables both (see Section 10.4). See Section 7.3.1 for further details.

### 7.2.15 Path Control Mode

The machining center may be put into any one of three path control modes: (1) exact stop mode, (2) exact path mode, or (3) continuous mode with optional tolerance. In exact stop mode, the machine stops briefly at the end of each programmed move. In exact path mode, the machine follows the programmed path as exactly as possible, slowing or stopping if necessary at sharp corners of the path. In continuous mode, sharp corners of the path may be rounded slightly so that the feed rate may be kept up (but by no more than the tolerance, if specified). See Section 9.15.

## 7.3 Interpreter Interaction with Switches

The Interpreter interacts with three switches. This section describes the interactions in more detail. In no case does the Interpreter know what the setting of any of these switches is.

### 7.3.1 Feed and Speed Override Switches

The Interpreter will interpret RS274/NGC commands which enable (M48) or disable (M49) the feed and speed override switches. It is useful to be able to override these switches for some machining operations. The idea is that optimal settings have been included in the program, and the operator should not change them.

EMC2 reacts to the setting of the speed or feed override switches on the control panel, when these switches are enabled.

### 7.3.2 Block Delete Switch

If the block delete switch is on, lines of RS274/NGC code which start with a slash (the block delete character) are not interpreted. If the switch is off, such lines are interpreted.

The Interpreter runs in two stages (read and execute). The driver tells the Interpreter when to perform each stage. When the Interpreter reads a line starting with a slash, it informs the driver, "I just read a line starting with a slash." The driver checks the setting of the block delete switch. If the switch is off, it tells the Interpreter, "Execute that line." If the switch is on, the driver does not tell the Interpreter to execute the line. Instead, it tells the Interpreter to read another line, with the result that the line starting with the slash is not executed.

### 7.3.3 Optional Program Stop Switch

The optional program stop switch works as follows. If this switch is on and an input RS274/NGC code line contains an M1 code, program execution is supposed to stop until the cycle start button is pushed.

EMC2 checks the optional stop switch when the `OPTIONAL_PROGRAM_STOP` canonical function call is executed and either stops (if the switch is on) or not (if the switch is off).

## 7.4 Tool File

A tool file is required to use the Interpreter. The file tells which tools are in which carousel slots and what the length and diameter of each tool are.

The format of a tool file is exemplified in Table 7.1.

Table 7.1: Sample Tool File

Pocket	FMS	TLO	Diameter	Comment
1	1	2.0	1.0	
2	2	1.0	0.2	
5	5	1.5	0.25	endmill
10	10	2.4	-0.3	for testing
21	21	173.740	0	1/2" spot drill
32	32	247.615	0	8.5mm drill
41	41	228.360	0	10mm tap
60	60	0	0	large chuck

The file consists of any number of header lines, followed by one blank line, followed by any number of lines of data. The header lines are ignored. It is important that there be exactly one blank line (with no spaces or tabs, even) before the data. The header line shown in Table 7.1 describes the data columns, so it is suggested (but not required) that such a line always be included in the header.

Each data line of the file contains the data for one tool. Each line has five entries. The first four entries are required. The last entry (a comment) is optional. It makes reading easier if the entries are arranged in columns, as shown in the table, but the only format requirement is that there be at least one space or tab after each of the first three entries on a line and a space, tab, or newline at the end of the fourth entry. The meanings of the columns and the type of data to be put in each are as follows.

The “Pocket” column contains an unsigned integer which represents the pocket number (slot number) of the tool carousel slot in which the tool is placed. The entries in this column must all be different.

The “FMS” column contains an unsigned integer which represents a code number for the tool. The user may use any code for any tool, as long as the codes are unsigned integers.

The “TLO” column contains a real number which represents the tool length offset. This number will be used if tool length offsets are being used and this pocket is selected. This is normally a positive real number, but it may be zero or any other number if it is never to be used.

The “Diameter” column contains a real number. This number is used only if tool radius compensation is turned on using this pocket. If the programmed path during compensation is the edge of the material being cut, this should be a positive real number representing the measured diameter of the tool. If the programmed path during compensation is the path of a tool whose diameter is nominal, this should be a small number (positive, negative, or zero) representing the difference between the measured diameter of the tool and the nominal diameter. If cutter radius compensation is not used with a tool, it does not matter what number is in this column.

The “Comment” column may optionally be used to describe the tool. Any type of description is OK. This column is for the benefit of human readers only.

The units used for the length and diameter of the tool may be in either millimeters or inches, but if the data is used by an NC program, the user must be sure the units used for a tool in the file are the same as the units in effect when NC code that uses the tool data is interpreted. The table shows a mixture of types of units.

The lines do not have to be in any particular order. Switching the order of lines has no effect unless the same slot number is used on two or more lines, which should not normally be done, in which case the data for only the last such line will be used.

## 7.5 Parameters

In the RS274/NGC language view, a machining center maintains an array of 5400 numerical parameters. Many of them have specific uses. The parameter array persists over time, even if the machining center is powered down. EMC2 uses a parameter file to ensure persistence and gives the Interpreter the responsibility for maintaining the file. The Interpreter reads the file when it starts up, and writes the file when it exits.

Table 7.2: Parameters Used by the RS274NGC Interpreter

Parameter Number(s)	Meaning
5161-5166	“G28” Home
5181-5186	“G30” Home
5211-5216	“G92” offset
5220	Coordinate System Number
5221-5226	Coordinate System 1
5241-5246	Coordinate System 2
5261-5266	Coordinate System 3
5281-5286	Coordinate System 4
5301-5306	Coordinate System 5
5321-5326	Coordinate System 6
5341-5346	Coordinate System 7
5361-5366	Coordinate System 8
5381-5386	Coordinate System 9

The format of a parameter file is shown in Table 7.3. The file consists of any number of header lines, followed by one blank line, followed by any number of lines of data. The Interpreter skips over the header lines. It is important that there be exactly one blank line (with no spaces or tabs, even) before the data. The header line shown in Table 7.3 describes the data columns, so it is suggested (but not required) that that line always be included in the header.

The Interpreter reads only the first two columns of the table. The third column, “Comment,” is not read by the Interpreter.

Each line of the file contains the index number of a parameter in the first column and the value to which that parameter should be set in the second column. The value is represented as a double-precision floating point number inside the Interpreter, but a decimal point is not required in the file. All of the parameters shown in Table 7.3 are required parameters and must be included in any parameter file, except that any parameter representing a rotational axis value for an unused axis may be omitted. An error will be signalled if any required parameter is missing. A parameter file may include any other parameter, as long as its number is in the range 1 to 5400. The parameter numbers must be arranged in ascending order. An error will be signalled if not. Any parameter included in the file read by the Interpreter will be included in the file it writes as it exits. The original file is saved as a backup file when the new file is written. Comments are not preserved when the file is written.

Table 7.3: Parameter File Format

Parameter Number	Parameter Value	Comment
5161	0.0	G28 Home X
5162	0.0	G28 Home Y

## 7.6 Coordinate Systems

In the RS274/NGC language view, a machining center has an absolute coordinate system and nine program coordinate systems.

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. See Section 9.6.

You can select one of the nine systems by using `G54`, `G55`, `G56`, `G57`, `G58`, `G59`, `G59.1`, `G59.2`, or `G59.3` (see Section 9.14). It is not possible to select the absolute coordinate system directly.

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 cancelled with `G92.1` or `G92.2`. See Section 9.18.

You can make straight moves in the absolute machine coordinate system by using `G53` with either `G0` or `G1`. See Section 9.13.

Data for coordinate systems is stored in parameters.

During initialization, the coordinate system is selected that is specified by parameter 5220. A value of 1 means the first coordinate system (the one `G54` activates), a value of 2 means the second coordinate system (the one `G55` activates), and so on. It is an error for the value of parameter 5220 to be anything but a whole number between one and nine.

# Chapter 8

## Language 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 General and 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 be 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 signalled 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. The interpreter does not even read them.

### 8.1 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, i.e., any letter outside a comment may be in upper or lower case without changing the meaning of a line.

## 8.2 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.

## 8.3 Word

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

Words may begin with any of the letters shown in Table 8.1. The table includes N for completeness, even though, as defined above, line numbers are not words. Several letters (I, J, K, L, P, R) may have different meanings in different contexts.

Table 8.1: Words and their meanings

Letter	Meaning
A	A axis of machine
B	B axis of machine
C	C axis of machine
D	Tool radius compensation number
F	Feedrate
G	General function (See table 5)
H	Tool length offset index
I	X offset for arcs and G87 canned cycles
J	Y offset for arcs and G87 canned cycles
K	Z offset for arcs and G87 canned cycles. Spindle-Motion Ratio for G33 synchronized movements.
M	Miscellaneous function (See table 7)
N	Line number
P	Dwell time in canned cycles and with G4. Key used with G10
Q	Feed increment in G83 canned cycle
R	Arc radius or canned cycle plane
S	Spindle speed
T	Tool selection
X	X axis of machine
Y	Y axis of machine
Z	Z axis of machine

### 8.3.1 Number

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 (1) an optional plus or minus sign, followed by (2) zero to many digits, followed, possibly, by (3) one decimal point, followed by (4) 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 to be close to an integer is considered close enough if it is within 0.0001 of an integer.

### 8.3.2 Parameter Value

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.

### 8.3.3 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 + acos[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.



### 8.3.4 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  $\text{FIX}[2.8] = 2$  and  $\text{FIX}[-2.8] = -3$ , for example. The FUP operation rounds towards the right (more positive or less negative) on a number line;  $\text{FUP}[2.8] = 3$  and  $\text{FUP}[-2.8] = -2$ , for example.

## 8.4 Parameter Setting

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.

## 8.5 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.

## 8.6 Repeated Items

A line may have any number of G words, but two G words from the same modal group (see Section 8.9) 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.

## 8.7 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.

## 8.8 Commands and Machine 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 cancelling motion.

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

## 8.9 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 are shown in Table 8.2.

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.

It is an error to include any unrelated words on a line with O- flow control.

Table 8.2: Modal Groups

Modal Group Meaning	Member Words
Motion ("Group 1")	G0 G1 G2 G3 G33 G38.2 G80 G81 G82 G83 G84 G85 G86 G87 G88 G89
Plane selection	G17 G18 G19
Distance Mode	G90 G91
Feed Rate Mode	G93, G94
Units	G20, G21
Cutter Radius Compensation	G40, G41, G42
Tool Length Offset	G43, G49
Return Mode in Canned Cycles	G98, G99
Coordinate System Selection	G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3
Stopping	M0, M1, M2, M30, M60
Tool Change	M6
Spindle Turning	M3, M4, M5
Coolant	M7, M8, M9. Special case: M7 and M8 may be active at the same time
Override Switches	M48, M49
Flow Control	O-
Non-modal codes ("Group 0")	G4, G10, G28, G30, G53 G92, G92.1, G92.2, G92.3 M100 to M199

# Chapter 9

## G Codes

G codes of the RS274/NGC language are shown in Table 5 and described following that.

In the command prototypes, the hyphen (-) stands for a real value. As described earlier, a real value may be (1) an explicit number, 4, for example, (2) an expression, [2+2], for example, (3) a parameter value, #88, for example, or (4) a unary function value, `acos[0]`, for example.

In most cases, if axis words (any or all of X-, Y-, Z-, A-, B-, C-) are given, they specify a destination point. Axis numbers are in the currently active coordinate system, unless explicitly described as being in the absolute coordinate system. Where axis words are optional, any omitted axes will have their current value. Any items in the command prototypes not explicitly described as optional are required. It is an error if a required item is omitted.

In the prototypes, the values following letters are often given as explicit numbers. Unless stated otherwise, the explicit numbers can be real values. For example, `G10 L2` could equally well be written `G[2*5] L[1+1]`. If the value of parameter 100 were 2, `G10 L#100` would also mean the same. Using real values which are not explicit numbers as just shown in the examples is rarely useful.

If L- is written in a prototype the “-” will often be referred to as the “L number”. Similarly the “-” in H- may be called the “H number”, and so on for any other letter.

### 9.1 G0: Rapid Linear Motion

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; see Chapter ???. If `G53` is programmed on the same line, the motion will also differ; see Section 9.13.

### 9.2 G1: 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; see Chapter ???. If G53 is programmed on the same line, the motion will also differ; see Section 9.13.

### 9.3 G2, G3: 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.

If a line of RS274/NGC code makes an arc and includes rotational axis motion, the rotational axes turn at a constant rate so that the rotational motion starts and finishes when the XYZ motion starts and finishes. Lines of this sort are hardly ever programmed.

If cutter radius compensation is active, the motion will differ from what is described here. See Chapter ??.

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.

#### 9.3.1 Radius format arcs

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.

#### 9.3.2 Center format arcs

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:  $\hat{A}$  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
- or 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,
- or 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
- or 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.

## 9.4 G33: Spindle-Synchronized Motion

For spindle-synchronized motion, code G33 X- Y- Z- K- where K gives the distance moved in XYZ for each revolution of the spindle. This syntax is subject to change (In particular, to use F- instead of K-). For instance, G33 Z1 K.0625 produces a 1 inch motion in Z over 16 revolutions of the spindle. This command might be part of a program to produce a 16TPI thread.

All the axis words are optional, except that at least one must be used. This will produce coordinated linear motion to the destination point at a rate dependant on the speed of the spindle.

It is an error if:

- all axis words are omitted.
- the spindle is not turning when this command is executed
- the requested linear motion exceeds machine velocity limits due to the spindle speed

## 9.5 G4: 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.

## 9.6 G10: Set Coordinate System Data

The RS274/NGC language view of coordinate systems is described in Section 7.6.

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.

## 9.7 G17, G18, G19: Plane Selection

Program G17 to select the XY-plane, G18 to select the XZ-plane, or G19 to select the YZ-plane. The effects of having a plane selected are discussed in Section 9.3 and Section 9.17

## 9.8 G20, G21: Length Units

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.

## 9.9 G28, G30: Return to 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.

## 9.10 G38.2: Straight Probe

This code is currently unimplemented in EMC2. It is accepted, but the behavior is undefined.

## 9.11 G40, G41, G42: Cutter Radius Compensation.

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

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,
- or cutter radius compensation is commanded to turn on when it is already on.

The behavior of the machining center when cutter radius compensation is on is described in Chapter ??

## 9.12 G43, G49: Tool Length Offsets

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.

To use no tool length offset, program G49.

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.

## 9.13 G53: Move in absolute coordinates

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,
- or G53 is used while cutter radius compensation is on.

See Section 7.6 for an overview of coordinate systems.

## 9.14 G54 to G59.3: Select Coordinate System

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.

See Section 7.6 for an overview of coordinate systems.

## 9.15 G61, G61.1, G64: Set Path Control Mode

Program G61 to put the machining center into exact path mode, G61.1 for exact stop mode, or G64 P- for continuous mode with optional tolerance. It is OK to program for the mode that is already active. See Section 7.2.15 for a discussion of these modes.

## 9.16 G80: Cancel Modal Motion

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.

## 9.17 G81 to G89: Canned Cycles

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. See Section 9.20

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,
- or 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,
- or 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,
- or 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,
- or the R number is less than the X number.

### 9.17.1 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.

### 9.17.2 G81: Drilling Cycle

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)

### 9.17.3 G82: Drilling Cycle 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.

### 9.17.4 G83: 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.

### 9.17.5 G84: Right-Hand Tapping

This code is currently unimplemented in EMC2. It is accepted, but the behavior is undefined.

### 9.17.6 G85: 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.

### 9.17.7 G86: 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.

### 9.17.8 G87: Back Boring

This code is currently unimplemented in EMC2. It is accepted, but the behavior is undefined.

### 9.17.9 G88: Boring, Spindle Stop, Manual Out

This code is currently unimplemented in EMC2. It is accepted, but the behavior is undefined.

### 9.17.10 G89: 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.

### 9.17.11 G90, G99: Set 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. Any exceptions to that rule are described explicitly in this Section 9.17.

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 Section 9.17.8), where the meaning changes with distance mode.

## 9.18 G92, G92.1, G92.2, G92.3: Coordinate System Offsets

See Section 7.6 for an overview of coordinate systems.

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:

1. 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  $[[7-9] + -3]$ .

To reset axis offsets to zero, program `G92.1` or `G92.2`. `G92.1` sets parameters 5211 to 5216 to zero, whereas `G92.2` leaves their current values alone.

To set the axis offset values to the values given in parameters 5211 to 5216, program `G92.3`.

You can set axis offsets in one program and use the same offsets in another program. Program `G92` in the first program. This will set parameters 5211 to 5216. Do not use `G92.1` in the remainder of the first program. The parameter values will be saved when the first program exits and restored when the second one starts up. Use `G92.3` near the beginning of the second program. That will restore the offsets saved in the first program. If other programs are to run between the the program that sets the offsets and the one that restores them, make a copy of the parameter file written by the first program and use it as the parameter file for the second program.

## 9.19 G93, G94: Set Feed Rate Mode

Two feed rate modes are recognized: units per minute and inverse time. Program `G94` to start the units per minute mode. Program `G93` to start the inverse time 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.

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.

## 9.20 G98, G99: Set Canned Cycle Return Level

When the spindle retracts during canned cycles, there is a choice of how far it retracts: (1) retract perpendicular to the selected plane to the position indicated by the R word, or (2) 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).

To use option (1), program G99. To use option (2), program G98. Remember that the R word has different meanings in absolute distance mode and incremental distance mode.

# Chapter 10

## M Codes

### 10.1 M0, M1, M2, M30, M60: Program Stopping and Ending

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

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

It is OK to program M0 and M1 in MDI mode, but the effect will probably not be noticeable, because normal behavior in MDI mode is to stop after each line of input, anyway.

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

If a program is stopped by an M0, M1, or M60, pressing the cycle start button will restart the program at the following line.

To end a program, program M2. To exchange pallet shuttles and then end a program, program M30. Both of these commands have the following effects.

1. Axis offsets are set to zero (like G92.2) and origin offsets are set to the default (like G54).
2. Selected plane is set to CANON\_PLANE\_XY (like G17).
3. Distance mode is set to MODE\_ABSOLUTE (like G90).
4. Feed rate mode is set to UNITS\_PER\_MINUTE (like G94).
5. Feed and speed overrides are set to ON (like M48).
6. Cutter compensation is turned off (like G40).
7. The spindle is stopped (like M5).
8. The current motion mode is set to G\_1 (like G1).
9. Coolant is turned off (like M9).

No more lines of code in an RS274/NGC file will be executed after the M2 or M30 command is executed. Pressing cycle start will start the program back at the beginning of the file.

### 10.2 M3, M4, M5: Spindle Control

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

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

To stop the spindle from turning, program M5.



It is OK to use M3 or M4 if the spindle speed is set to zero. If this is done (or if the speed override switch is enabled and set to zero), the spindle will not start turning. If, later, the spindle speed is set above zero (or the override switch is turned up), the spindle will start turning. It is OK to use M3 or M4 when the spindle is already turning or to use M5 when the spindle is already stopped.

### 10.3 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 - see Section 12.3), program M6. When the tool change is complete:

- The spindle will be stopped.
- The tool that was selected (by a T word on the same line or on any line after the previous tool change) will be in the spindle. The T number is an integer giving the changer slot of the tool (not its id).
- If the selected tool was not in the spindle before the tool change, the tool that was in the spindle (if there was one) will be in its changer slot.
- The coordinate axes will be stopped in the same absolute position they were in before the tool change (but the spindle may be re-oriented).
- No other changes will be made. For example, coolant will continue to flow during the tool change unless it has been turned off by an M9.

The tool change may include axis motion while it is in progress. It is OK (but not useful) to program a change to the tool already in the spindle. It is OK if there is no tool in the selected slot; in that case, the spindle will be empty after the tool change. If slot zero was last selected, there will definitely be no tool in the spindle after a tool change.

### 10.4 M7, M8, M9: Coolant Control

To turn mist coolant on, program M7.

To turn flood coolant on, program M8.

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.

### 10.5 M48, M49: Override Control

To enable the speed and feed override switches, program M48. To disable both switches, program M49. See Section 7.3.1 for more details. It is OK to enable or disable the switches when they are already enabled or disabled.

### 10.6 M100 to M199: User Defined Commands

To invoke a user-defined command, program M- P- Q- where P- and Q- are both optional. The external program “Mnnn” in the directory [DISPLAY]PROGRAM\_PREFIX is executed with the P and Q values as its two arguments. Execution of the RS274NGC file pauses until the invoked program exits.

It is an error if

- The specified User Defined Command does not exist

# Chapter 11

## O Codes

O-codes provide for flow control in NC programs. Each block has an associated number, which is the number used after O. Care must be taken to properly match the O-numbers.

### 11.1 Subroutines: “sub”, “endsub”, “return”, “call”

Subroutines extend from a O- sub to an O- endsub. The lines inside the subroutine (the “body”) are not executed in order; instead, they are executed each time the subroutine is called with O-call.

```
O100 sub (subroutine to move to machine home)
G0 X0 Y0 Z0
O100 endsub
(many intervening lines)
O100 call
```

Inside a subroutine, O- return can be executed. This immediately returns to the calling code, just as though O- endsub was encountered.

O- call takes optional arguments, which are passed to the subroutine as #1, #2, etc up to #30. On return from the subroutine the original values of those parameters will be restored.

Subroutine calls may not be nested. They may only be called after they are defined. They may be called from other functions, and may call themselves recursively if it makes sense to do so. The maximum subroutine nesting level is 10.

### 11.2 Looping: “do”, “while”, “endwhile”, “break”, “continue”

The “while loop” has two structures: while/endwhile, and do/while. In each case, the loop is exited when the “while” condition is false.

```
(draw a sawtooth shape)
F100
#1 = 0
O101 while [#1 lt 10]
G1 X0
G1 Y[#1/10] X1
#1 = [#1+1]
O101 endwhile
```

Inside a while loop, `O- break` immediately exits the loop, and `O- continue` immediately skips to the next evaluation of the `while` condition. If it is still true, the loop begins again at the top. If it is false, it exits the loop.

### 11.3 Conditional: “if”, “else”, “endif”

The “if” conditional executes one group of statements if a condition is true and another if it is false.

```
(Set feed rate depending on a variable)
O102 if [#2 > 5]
F100
O102 else
F200
O102 endif
```

# Chapter 12

## Other Codes

### 12.1 F: Set Feed Rate

To set the feed rate, program `F-`. The application of the feed rate is as described in Section 7.2.5, unless inverse time feed rate mode is in effect, in which case the feed rate is as described in Section 9.19.

### 12.2 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.

As described in Section 9.17.5, if a `G84` (tapping) canned cycle is active and the feed and speed override switches are enabled, the one set at the lower setting will take effect. The speed and feed rates will still be synchronized. In this case, the speed may differ from what is programmed, even if the speed override switch is set at 100%.

### 12.3 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 (see Section 10.3). 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,
- or a T number larger than the number of slots in the carousel is used.

On some machines, the carousel will move when a T word is programmed, at the same time machining is occurring. On such machines, programming the T word several lines before a tool change will save time. A common programming practice for such machines is to put the T word for the next tool to be used on the line after a tool change. This maximizes the time available for the carousel to move.

# Chapter 13

## Order of Execution

The order of execution of items on a line is critical to safe and effective machine operation. Items are executed in the order shown below if they occur on the same line.

1. Comment (including message)
2. set feed rate mode (G93, G94).
3. set feed rate (F).
4. set spindle speed (S).
5. select tool (T).
6. change tool (M6).
7. spindle on or off (M3, M4, M5).
8. coolant on or off (M7, M8, M9).
9. . enable or disable overrides (M48, M49).
10. dwell (G4).
11. set active plane (G17, G18, G19).
12. set length units (G20, G21).
13. cutter radius compensation on or off (G40, G41, G42)
14. cutter length compensation on or off (G43, G49)
15. coordinate system selection (G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3).
16. set path control mode (G61, G61.1, G64)
17. set distance mode (G90, G91).
18. set retract mode (G98, G99).
19. home (G28, G30) or change coordinate system data (G10) or set axis offsets (G92, G92.1, G92.2, G94).
20. perform motion (G0 to G3, G33, G80 to G89), as modified (possibly) by G53.
21. stop (M0, M1, M2, M30, M60).

# Chapter 14

## G-Code Best Practices

### 14.1 Use an appropriate decimal precision

Use 3 digits after the decimal when milling in inches, and 4 digits after the decimal when milling in millimeters. In particular, arc tolerance checks are made to .001 and .0001 depending on the active units.

### 14.2 Use consistent white space

G-code is most legible when at least one space appears before words. While it is permitted to insert whitespace in the middle of numbers, there is no reason to do so.

### 14.3 Prefer “Center-format” arcs

Center-format arcs (which use I- J- K- instead of R-) behave more consistently than R-format arcs, particularly for included angles near 180 degrees.

### 14.4 Put important modal settings at the top of the file

When correct execution of your program depends on modal settings, set them once and at the beginning of the file. Perhaps the most critical modal setting is the distance value–If you do not include G20 or G21, then different machines will mill the program at different scales. Other settings, such as the return mode in canned cycles may also be important.

### 14.5 Don’t put too many things on one line

Ignore everything in Section 13, and instead write no line of code that is the slightest bit ambiguous. Similarly, don’t use and set a parameter on the same line, even though the semantics are well defined. (Exception: Updating a variable to a new value, such as #1=#1+#2)

### 14.6 Don’t use line numbers

Line numbers offer no benefits. When line numbers are reported in error messages, the numbers refer to the physical line number, not the N-word value.

# Chapter 15

## Tool File and Compensation

### 15.1 Tool File

The EMC uses a tool file that is read in when a machine control is started. In a standard release this file is named *emc.var*, *generic.var*, or *sim.var* and is used by the similarly named *run* file. The specific name of the file that will be used is set by the *ini file* that is read at startup.

A tool file is required. It tells which tools are in which carousel slots and what the length and diameter of each tool are. The Interpreter does not deal directly with tool files. A tool file is read by the EMC system and the Interpreter gets the tool information by making calls to canonical functions that obtain it.

The header line shown in Table 15.1 is essential for some of the graphical interfaces, so it is suggested (but not required) that such a line always be included as the first line in the file.

Figure 15.1: Typical Tool File

POC	FMS	LEN	DIAM	COMMENT
1	1	1.565	0.250	Drill
2	2	1.000	0.247	Reground End Mill
3	3	1.125	2.000	Carbide Insert Face Mill
4	4	-	-	-
5	5	-	-	-
32	32	-	-	-

Each data line of the file contains the data for one tool. Each line has five entries. The first four entries are required. The last entry (a comment) is optional. It makes reading easier if the entries are arranged in columns, as shown in the table, but the only format requirement is that there be at least one space or tab after each of the first three entries on a line and a space, tab, or newline at the end of the fourth entry. The meanings of the

columns and the type of data to be put in each are as follows.

The "POC" column contains an unsigned integer which represents the pocket number (slot number) of the tool carousel slot in which the tool is placed. The entries in this column must all be different.

The "FMS" column contains an unsigned integer which represents a code number for the tool. The user may use any code for any tool, as long as the codes are unsigned integers.

The "LEN" column contains a real number which represents the tool length offset. This number will be used if tool length offsets are being used and this pocket is selected. This is normally a positive real number, but it may be zero.

The "DIAM" column contains a real number. This number is used if tool radius compensation is turned on using this pocket number. If the programmed path during compensation is the edge of the material being cut, this should be a positive real number representing the measured diameter of the tool. If the programmed path during compensation is the path of a tool whose diameter is nominal, this should be a small number (positive, negative, or zero) representing the difference between the measured diameter of the tool and the nominal diameter used when the G-code for the part was written.

The "COMMENT" column may optionally be used to describe the tool. Any type of description is OK. This column is for the benefit of human readers only.

The units used for the length and diameter of the tool may be in either millimeters or inches, but if the data is used by an NC program, the user must be sure the units used for a tool in the file are the same as the units in effect when NC code that uses the tool data is interpreted.

The lines do not have to be in any particular order. Switching the order of lines has no effect. If the same pocket number is used on two or more lines, which should not normally be done, the data for only the last such line will persist and be used.

## 15.2 Tool Compensation

Tool compensation can cause problems for the best of nc code programmers. But it can be a powerful aid when used to help an operator get a part to size. By setting and resetting length and diameter of tools in a single tool table, offsets can be made during a production run that allow for variation in tool size, or for minor deviation from the programmed distances and size. And these changes can be made without the operator having to search through and change numbers in a program file.

Throughout this unit you will find occasional references to canonical functions where these are necessary for the reader to understand how a tool offset works in a specific situation. These references are intended to give the reader a sense of sequence rather than requiring the reader to understand the way that canonical functions themselves work within the EMC.

## 15.3 Tool Length Offsets

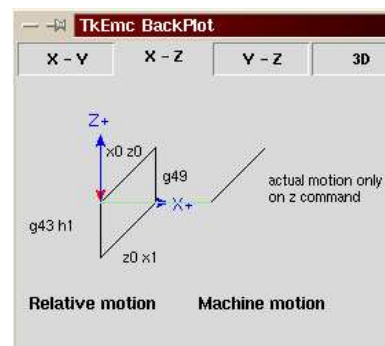
Tool length offsets are given as positive numbers in the tool table. A tool length offset is programmed using G43 Hn, where n is the desired table index. It is expected that all entries in the tool table will be positive. The H number is checked for being a non-negative integer when it is read. The interpreter behaves as follows.

1. If G43 Hn is programmed, A USE\_TOOL\_LENGTH\_OFFSET(length) function call is made (where length is the value of the tool length offset entry in the tool table whose index is n), tool\_length\_offset is reset in the machine settings model, and the value of current\_z in the model is adjusted. Note that n does not have to be the same as the slot number of the tool currently in the spindle.
2. If G49 is programmed, USE\_TOOL\_LENGTH\_OFFSET(0.0) is called, tool\_length\_offset is reset to 0.0 in the machine settings model, and the value of current\_z in the model is adjusted. The effect of tool length compensation is illustrated in the screen shot below. Notice that the length of the tool is subtracted from the z setting so that the tool tip appears at the programmed setting. You should note that the effect of tool length compensation is immediate when you view the z position as a relative coordinate but it does affect actual machine position until you program a z move.

Test tool length program.

Tool #1 is 1 inch long.

```
N01 G1 F15 X0 Y0 Z0
N02 G43 H1 Z0 X1
N03 G49 X0 Z0
N04 G0 X2
N05 G1 G43 H1 G4 P10 Z0 X3
N06 G49 X2 Z0
N07 G0 X0
```



The effect of this is that in most cases the machine will pick up the offset as a ramp during the next xyz move after the g43 word.



## 15.4 Cutter Radius Compensation

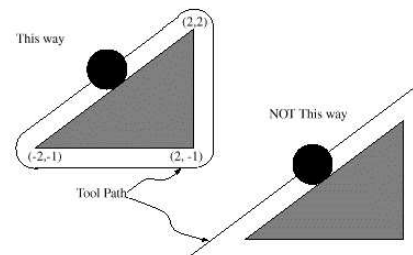
Cutter Diameter Compensation (also called Cutter Radius Compensation) is something that was obviously added onto the RS-274D specification at the demand of users, as it is VERY useful, but the implementation was poorly thought out. The purpose of this feature is to allow the programmer of the tool path program to 'virtualize' the tool path, so that the control can, at run time, determine the correct offset from the surface to be cut, based on the tools available. If you resharpen the side cutting edges of end mills, then they will end up smaller than the standard diameters.

The problem is to describe to the control whether the tool is going to be cutting on the outside of an imaginary path, or on the inside. Since these paths are not necessarily closed paths (although they can be), it is essentially impossible for the control to know which side of the line it is supposed to offset to. It was decided that there would only be two choices, tool 'left' of path, and tool 'right' of path. This is to be interpreted as left or right 'when facing the direction of cutter motion'. The interpretation is as if you were standing on the part, walking behind the tool as it progresses across the part.

### 15.4.1 Cutter Radius Compensation Detail

The cutter radius compensation capabilities of the interpreter enable the programmer to specify that a cutter should travel to the right or left of an open or closed contour in the XY-plane composed of arcs of circles and straight line segments. The contour may be the outline of material not to be machined away, or it may be a tool path to be followed by an exactly sized tool. This figure shows two examples of the path of a tool cutting using cutter radius compensation so that it leaves a triangle of material remaining.

In both examples, the shaded triangle represents material which should remain after cutting, and the line outside the shaded triangle represents the path of the tip of a cutting tool. Both paths will leave the shaded triangle uncut. The one on the left (with rounded corners) is the path the interpreter will generate. In the method on the right (the one not used), the tool does not stay in contact with the shaded triangle at sharp corners.



Z axis motion may take place while the contour is being followed in the XY plane. Portions of the contour may be skipped by retracting the Z axis above the part, following the contour to the next point at which machining should be done, and re-extending the Z-axis. These skip motions may be performed at feed rate (G1) or at traverse rate (G0). Inverse time feed rate (G93) or units per minute feed rate (G94) may be used with cutter radius compensation. Under G94, the feed rate will apply to the actual path of the cutter tip, not to the programmed contour.

### Programming Instructions

- To start cutter radius compensation, program either G41 (for keeping the tool to the left of the contour) or G42 (for keeping the tool to the right of the contour). In Figure 7, for example, if G41 were programmed, the tool would stay left and move clockwise around the triangle, and if G42 were programmed, the tool would stay right and move counterclockwise around the triangle.
- To stop cutter radius compensation, program G40.
- If G40, G41, or G42 is programmed in the same block as tool motion, cutter compensation will be turned on or off before the motion is made. To make the motion come first, the motion must be programmed in a separate, previous block.

## D Number

The current interpreter requires a D number on each line that has the G41 or G42 word. The value specified with D must be a non-negative integer. It represents the slot number of the tool whose radius (half the diameter given in the tool table) will be used, or it may be zero (which is not a slot number). If it is zero, the value of the radius will also be zero. Any slot in the tool table may be selected this way. The D number does not have to be the same as the slot number of the tool in the spindle.

## Tool Table

Cutter radius compensation uses data from the machining center's tool table. For each slot in the tool carousel, the tool table contains the diameter of the tool in that slot (or the difference between the actual diameter of the tool in the slot and its nominal value). The tool table is indexed by slot number. How to put data into the table when using the stand-alone interpreter is discussed in the tool table page.

## Two Kinds of Contour

The interpreter handles compensation for two types of contour:

- The contour given in the NC code is the edge of material that is not to be machined away. We will call this type a "material edge contour".
- The contour given in the NC code is the tool path that would be followed by a tool of exactly the correct radius. We will call this type a "tool path contour".

The interpreter does not have any setting that determines which type of contour is used, but the description of the contour will differ (for the same part geometry) between the two types and the values for diameters in the tool table will be different for the two types.

## Material Edge Contour

When the contour is the edge of the material, the outline of the edge is described in the NC program. For a material edge contour, the value for the diameter in the tool table is the actual value of the diameter of the tool. The value in the table must be positive. The NC code for a material edge contour is the same regardless of the (actual or intended) diameter of the tool.

### Example 1 :

Here is an NC program which cuts material away from the outside of the triangle in figure above. In this example, the cutter compensation radius is the actual radius of the tool in use, which is 0.5. The value for the diameter in the tool table is twice the radius, which is 1.0.

```
N0010 G41 G1 X2 Y2 (turn compensation on and make entry move)
N0020 Y-1 (follow right side of triangle)
N0030 X-2 (follow bottom side of triangle)
N0040 X2 Y2 (follow hypotenuse of triangle)
N0050 G40 (turn compensation off)
```

This will result in the tool following a path consisting of an entry move and the path shown on the left going clockwise around the triangle. Notice that the coordinates of the triangle of material appear in the NC code. Notice also that the tool path includes three arcs which are not explicitly programmed; they are generated automatically.

### Tool Path Contour

When the contour is a tool path contour, the path is described in the NC program. It is expected that (except for during the entry moves) the path is intended to create some part geometry. The path may be generated manually or by a post-processor, considering the part geometry which is intended to be made. For the interpreter to work, the tool path must be such that the tool stays in contact with the edge of the part geometry, as shown on the left side of Figure 7. If a path of the sort shown on the right of Figure 7 is used, in which the tool does not stay in contact with the part geometry all the time, the interpreter will not be able to compensate properly when undersized tools are used.

For a tool path contour, the value for the cutter diameter in the tool table will be a small positive number if the selected tool is slightly oversized and will be a small negative number if the tool is slightly undersized. As implemented, if a cutter diameter value is negative, the interpreter compensates on the other side of the contour from the one programmed and uses the absolute value of the given diameter. If the actual tool is the correct size, the value in the table should be zero.

#### Tool Path Contour example

Suppose the diameter of the cutter currently in the spindle is 0.97, and the diameter assumed in generating the tool path was 1.0. Then the value in the tool table for the diameter for this tool should be -0.03. Here is an NC program which cuts material away from the outside of the triangle in the figure.

```
N0010 G1 X1 Y4.5 (make alignment move)
N0020 G41 G1 Y3.5 (turn compensation on and make first entry move)
N0030 G3 X2 Y2.5 I1 (make second entry move)
N0040 G2 X2.5 Y2 J-0.5 (cut along arc at top of tool path)
N0050 G1 Y-1 (cut along right side of tool path)
N0060 G2 X2 Y-1.5 I-0.5 (cut along arc at bottom right of tool path)
N0070 G1 X-2 (cut along bottom side of tool path)
N0080 G2 X-2.3 Y-0.6 J0.5 (cut along arc at bottom left of tool path)
N0090 G1 X1.7 Y2.4 (cut along hypotenuse of tool path)
N0100 G2 X2 Y2.5 I0.3 J-0.4 (cut along arc at top of tool path)
N0110 G40 (turn compensation off)
```

This will result in the tool making an alignment move and two entry moves, and then following a path slightly inside the path shown on the left in Figure 7 going clockwise around the triangle. This path is to the right of the programmed path even though G41 was programmed, because the diameter value is negative.

### Programming Errors and Limitations

The interpreter will issue the following messages involving cutter radius compensation.

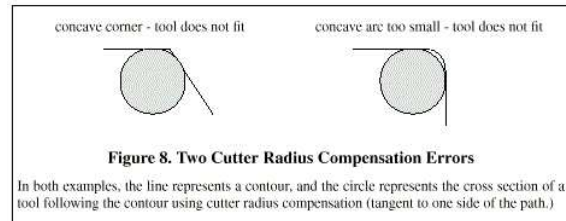
1. Cannot change axis offsets with cutter radius comp
2. Cannot change units with cutter radius comp
3. Cannot turn cutter radius comp on out of XY-plane
4. Cannot turn cutter radius comp on when already on
5. Cannot use G28 or G30 with cutter radius comp
6. Cannot use G53 with cutter radius comp
7. Cannot use XZ plane with cutter radius comp
8. Cannot use YZ plane with cutter radius comp
9. Concave corner with cutter radius comp
10. Cutter gouging with cutter radius comp
11. D word on line with no cutter comp on (G41 or G42) command

12. Tool radius index too big
13. Tool radius not less than arc radius with cutter radius comp
14. Two G codes used from same modal group.

For some of these messages additional explanation is given below.

Changing a tool while cutter radius compensation is on is not treated as an error, although it is unlikely this would be done intentionally. The radius used when cutter radius compensation was first turned on will continue to be used until compensation is turned off, even though a new tool is actually being used.

When cutter radius compensation is on, it must be physically possible for a circle whose radius is the half the diameter given in the tool table to be tangent to the contour at all points of the contour.



In particular, the interpreter treats concave corners and concave arcs into which the circle will not fit as errors, since the circle cannot be kept tangent to the contour in these situations. This error detection does not limit the shapes which can be cut, but it does require that the programmer specify the actual shape to be cut (or path to be followed), not an approximation. In this respect, the interpreter differs from interpreters used with many other controllers, which often allow these errors silently and either gouge the part or round the corner. If cutter radius compensation has already been turned on, it cannot be turned on again. It must be turned off first; then it can be turned on again. It is not necessary to move the cutter between turning compensation off and back on, but the move after turning it back on will be treated as a first move, as described below.

It is not possible to change from one cutter radius index to another while compensation is on because of the combined effect of rules 4 and 11. It is also not possible to switch compensation from one side to another while compensation is on. If the tool is already covering up the next XY destination point when cutter radius compensation is turned on, the gouging message is given when the line of NC code which gives the point is reached. In this situation, the tool is already cutting into material it should not cut.

If a D word is programmed that is larger than the number of tool carousel slots, an error message is given. In the current implementation, the number of slots is 68.

The error message, "two G Codes Used from Same Modal Group," is a generic message used for many sets of G codes. As applied to cutter radius compensation, it means that more than one of G40, G41, and G42 appears on a line of NC code. This is not allowed.

## First Move

The algorithm used for the first move when the first move is a straight line is to draw a straight line from the destination point which is tangent to a circle whose center is at the current point and whose radius is the radius of the tool. The destination point of the tool tip is then found as the center of a circle of the same radius tangent to the tangent line at the destination point. This is shown in Figure 9. If the programmed point is inside the initial cross section of the tool (the circle on the left), an error is signalled.

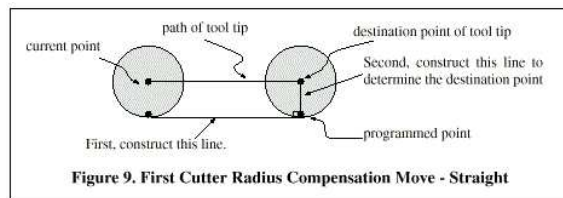


Figure 9. First Cutter Radius Compensation Move - Straight

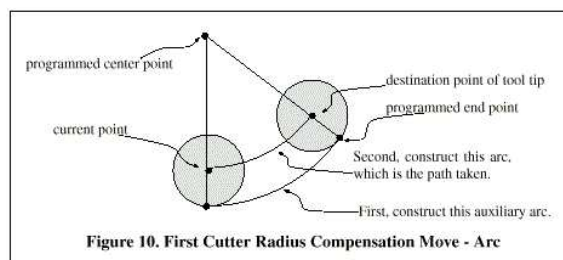


Figure 10. First Cutter Radius Compensation Move - Arc

If the first move after cutter radius compensation has been turned on is an arc, the arc which is generated is derived from an auxiliary arc which has its center at the programmed center point, passes through the programmed end point, and is tangent to the cutter at its current location. If the auxiliary arc cannot be constructed, an error is signalled. The generated arc moves the tool so that it stays tangent to the auxiliary arc throughout the move. This is shown in Figure 10.

Regardless of whether the first move is a straight line or an arc, the Z axis may also move at the same time. It will move linearly, as it does when cutter radius compensation is not being used. Rotary axis motions (A, B, and C axes) are allowed with cutter radius compensation, but using them would be very unusual.

After the entry moves of cutter radius compensation, the interpreter keeps the tool tangent to the programmed path on the appropriate side. If a convex corner is on the path, an arc is inserted to go around the corner. The radius of the arc is half the diameter given in the tool table.

When cutter radius compensation is turned off, no special exit move takes place. The next move is what it would have been if cutter radius compensation had never been turned on and the previous move had placed the tool at its current position.

## Programming Entry Moves

In general, an alignment move and two entry moves are needed to begin compensation correctly. However, where the programmed contour is a material edge contour and there is a convex corner on the contour, only one entry move (plus, possibly, a pre-entry move) is needed. The general method, which will work in all situations, is described first. We assume here that the programmer knows what the contour is already and has the job of adding entry moves.

### General Method

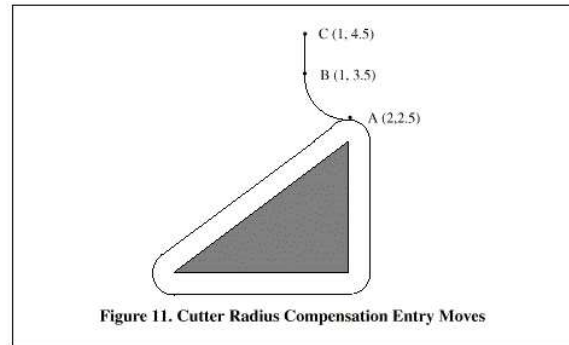
The general method includes programming an alignment move and two entry moves. The entry moves given above will be used as an example. Here is the relevant code again:

```
N0010 G1 X1 Y4.5 (make alignment move to point C)
N0020 G41 G1 Y3.5 (turn compensation on and make first entry move to point B)
N0030 G3 X2 Y2.5 I1 (make second entry move to point A)
```

See Figure 11. The figure shows the two entry moves but not the alignment move.

First, pick a point A on the contour where it is convenient to attach an entry arc. Specify an arc outside the contour which begins at a point B and ends at A tangent to the contour (and going in the same direction as it is planned to go around the contour). The radius of the arc should be larger than half the diameter given in the tool table. Then extend a line tangent to the arc from B to some point C, located so that the line BC is more than one radius long.

After the construction is finished, the code is written in the reverse order from the construction. Cutter radius compensation is turned on after the alignment move and before the first entry move. In the code above, line N0010 is the alignment move, line N0020 turns compensation on and makes the first entry move, and line N0030 makes the second entry move.



In this example, the arc AB and the line BC are fairly large, but they need not be. For a tool path contour, the radius of arc AB need only be slightly larger than the maximum possible deviation of the radius of the tool from the exact size. Also for a tool path contour, the side chosen for compensation should be the one to use if the tool is oversized. As mentioned earlier, if the tool is undersized, the interpreter will switch sides.

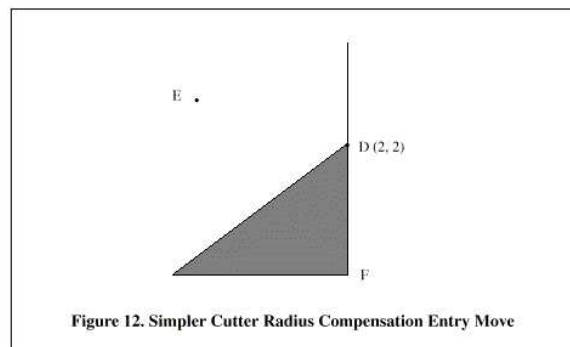
### Simple Method

If the contour is a material edge contour and there is a convex corner somewhere on the contour, a simpler method of making an entry is available. See Figure 12.

First, pick a convex corner, D. Decide which way you want to go along the contour from D. In our example we are keeping the tool to the left of the contour and going next towards F. Extend the line FD (if the next part of the contour is an arc, extend the tangent to arc FD from D) to divide the area outside the contour near D into two regions. Make sure the center of the tool is currently in the region on the same side of the extended line as the material inside the contour near D. If not, move the tool into that region. In the example, point E represents the current location of the center of the tool. Since it is on the same side of line DF as the shaded triangle, no additional move is needed. Now write a line of NC code that turns compensation on and moves to point D

```
N0010 G41 G1 X2 Y2 (turn
compensation on and make entry
move)
```

This method will also work at a concave corner on a tool path contour, if the actual tool is oversized, but it will fail with a tool path contour if the tool is undersized.



### Other Items Where Cutter Radius Compensation is Performed.

The complete set of canonical functions includes functions which turn cutter radius on and off, so that cutter radius compensation can be performed in the controller executing the canonical

functions. In the interpreter, however, these commands are not used. Compensation is done by the interpreter and reflected in the output commands, which continue to direct the motion of the center of the cutter tip. This simplifies the job of the motion controller while making the job of the interpreter a little harder.

### Algorithms for Cutter Radius Compensation

The interpreter allows the entry and exit moves to be arcs. The behavior for the intermediate moves is the same, except that some situations treated as errors in the interpreter are not treated as errors in other machine controls.

#### Data for Cutter Radius Compensation

The interpreter machine model keeps three data items for cutter radius compensation: the setting itself (right, left, or off), program\_x, and program\_y. The last two represent the X and Y positions which are given in the NC code while compensation is on. When compensation is off, these both are set to a very small number (10<sup>-20</sup>) whose symbolic value (in a #define) is "unknown". The interpreter machine model uses the data items current\_x and current\_y to represent the position of the center of the tool tip (in the currently active coordinate system) at all times.

### Jon Elson's Example

All further system-specific information refers to NIST's EMC program, but much of it applies to most modern CNC controls. My method of checking these programs is to first select tool zero, which always has a diameter of zero, so offset commands are essentially ignored. Then, I tape a sheet of paper to a piece of material that sits level in my vise, as a sort of platen. I install a spring-loaded pen in the spindle. This is a standard ballpoint pen refill cartridge made of metal, in a 1/2" diameter steel housing. It has a spring that loads the pen against the front, and a 'collet' at the front that allows the pen to retract against the spring, but keeps it centered within a few thousandths of an inch. I run the program with tool zero selected, and it draws a line at the actual part's outline. (see figure below) Then, I select a tool with the diameter of the tool I intend to use, and run the program again. (Note that Z coordinates in the program may need to be changed to prevent plunging the pen through the platen.) Now, I get to see whether the G41 or G42 compensation that I specified will cut on the desired side of the part. If it doesn't, I then edit the opposite compensation command into the program, and try again. Now, with the tool on the correct side of the work, you get to see if there are any places where the tool is 'too fat' to fit in a concave part of the surface. My old Allen-Bradley 7320 was pretty forgiving on this, but EMC is a complete stickler. If you have ANY concavity where two lines meet at less than 180 degrees on the side that a tool of finite size cuts on, EMC will stop with an error message there. Even if the gouge will be .0001" deep. So, I always make the approach on the lead-in and lead-out moves such that they just nip the corner of the part a tiny bit, providing an angle just over 180 degrees, so that EMC won't squawk. This requires some careful adjustment of the starting and ending points, which are not compensated by cutter radius, but must be chosen with an approximate radius in mind.

The operative commands are :

G40 - Cancel Cutter compensation

G41 - Cutter Compensation, Tool Left of Path

G42 - Cutter Compensation, Tool Right of Path

Here is a short file that cuts one side of a part with multiple convex and concave arcs, and several straight cuts, too. It is to clamp a high speed drilling spindle to the side of the main Bridgeport spindle. Most of these commands are straight from Bobcad/CAM, but lines N15 and N110 were added by me, and some of the coordinates around those lines had to be fudged a bit by me.

```
N10 G01 G40 X-1.3531 Y3.4
N15 F10 G17 G41 D4 X-0.7 Y3.1875 (COMP LEAD IN)
N20 X0. Y3.1875
N40 X0.5667 F10
N50 G03 X0.8225 Y3.3307 R0.3
N60 G02 X2.9728 Y4.3563 R2.1875
```

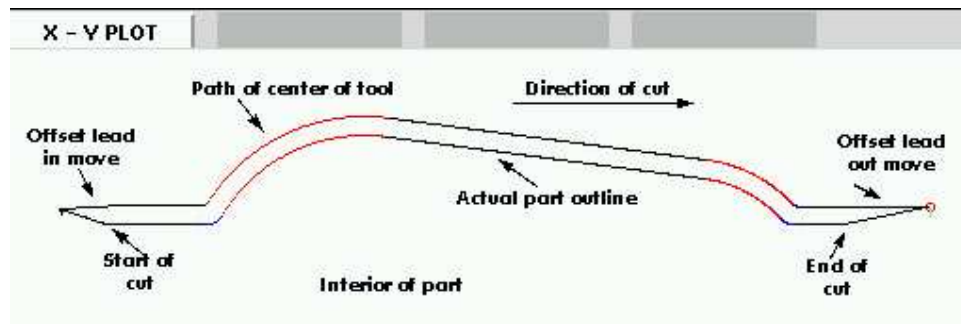
```

N70 G01 X7.212 Y3.7986
N80 G02 X8.1985 Y3.2849 R1.625
N90 G03 X8.4197 Y3.1875 R0.3
N100 G01 X9.
N110 G40 X10.1972 Y3.432 (COMP LEAD OUT
N220 M02

```

Line 15 contains G41 D4, which means that the diameter of the tool described as tool #4 in the tool table will be used to offset the spindle by 1/2 the diameter, which is, of course, the tool's radius. Note that the line with the G41 command contains the endpoint of the move where the radius compensation is interpolated in. What this means is that at the beginning of this move, there is no compensation in effect, and at the end, the tool is offset by 100% of the selected tool radius. Immediately after the G41 is D4, meaning that the offset is by the radius of tool number 4 in the tool table. Note that tool DIAMETERS are entered in the tool table. (Jon's tool diameter is about 0.4890)

But, note that in line 110, where the G40 'cancel cutter compensation' command is, that cutter compensation will be interpolated out in this move. The way I have these set up, the moves in lines 15 and 110 are almost exactly parallel to the X axis, and the difference in Y coordinates is to line the tool up outside the portion of the program where cutter compensation is in force.



Some other things to note are that the program starts with a G40, to turn off any compensation that was in effect. This saves a lot of hassle when the program stops due to a concavity error, but leaves the compensation turned on. Also note in line 15 that G17 is used to select the XY plane for circular interpolation. I have used the radius form of arc center specification rather than the I,J form. EMC is very picky about the radius it computes from I,J coordinates, and they must match at the beginning and end of the move to within  $10^{-11}$  internal units, so you will have lots of problems with arbitrary arcs. Usually, if you do an arc of 90 degrees, centered at (1.0,1.0) with a radius of 1", everything will go fine, but if it has a radius that can not be expressed exactly in just a few significant digits, or the arc is a strange number of degrees, then there will be trouble with EMC. The R word clears up all that mess, and is a lot easier to work with, anyway. If the arc is more than 180 degrees, R should be negative.

## 15.5 Tool Compensation Sources

This unit borrows heavily from the published works of Tom Kramer and Fred Proctor at NIST and the cutter compensation web page of Jon Elson.

Papers by Tom Kramer and Fred Proctor

<http://www.isd.mel.nist.gov/personnel/kramer/publications.html>

[http://www.isd.mel.nist.gov/personnel/kramer/pubs/RS274NGC\\_22.pdf](http://www.isd.mel.nist.gov/personnel/kramer/pubs/RS274NGC_22.pdf)

[http://www.isd.mel.nist.gov/personnel/kramer/pubs/RS274VGER\\_11.pdf](http://www.isd.mel.nist.gov/personnel/kramer/pubs/RS274VGER_11.pdf)

Pages by Jon Elson

<http://artsci.wustl.edu/~jmelson/>

<http://206.19.206.56/diacomp.htm>

<http://206.19.206.56/lencomp.htm>





# Chapter 16

## Coordinate System and G92 Offsets

### 16.1 Introduction

You have seen how handy a tool length offset can be. Having this allows the programmer to ignore the actual tool length when writing a part program. In the same way, it is really nice to be able to find a prominent part of a casting or block of material and work a program from that point rather than having to take account of the location at which the casting or block will be held during the machining.

This chapter introduces you to offsets as they are used by the EMC. These include;

- machine coordinates (G53)
- nine offsets (G54-G59.3)
- a set of global offsets (G92).

### 16.2 The Machine Position Command (G53)

Regardless of any offsets that may be in effect, putting a G53 in a block of code tells the interpreter to go to the real or absolute axis positions commanded in the block. For example

```
g53 g0 x0 y0 z0
```

will get you to the actual position where these three axes are zero. You might use a command like this if you have a favorite position for tool changes or if your machine has an auto tool changer. You might also use this command to get the tool out of the way so that you can rotate or change a part in a vice.

G53 is not a modal command. It must be used on each line where motion based upon absolute machine position is desired.

### 16.3 Fixture Offsets (G54-G59.3)

Work or fixture offset are used to make a part home that is different from the absolute, machine coordinate system. This allows the part programmer to set up home positions for multiple parts. A typical operation that uses fixture offsets would be to mill multiple copies of parts on "islands" in a piece, similar to figure 16.1

The values for offsets are stored in the VAR file that is requested by the INI file during the startup of an EMC. In our example below we'll use G55. The values for each axis for G55 are stored as variable numbers.

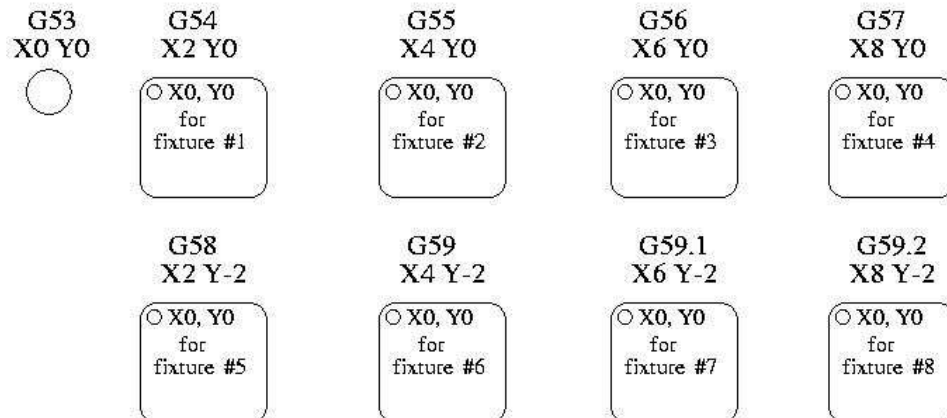


Figure 16.1: Work Offsets

```

5241  0.000000
5242  0.000000
5243  0.000000
5244  0.000000
5245  0.000000
5246  0.000000

```

In the VAR file scheme, the first variable number stores the X offset, the second the Y offset and so on for all six axes. There are numbered sets like this for each of the fixture offsets.

Each of the graphical interfaces has a way to set values for these offsets. You can also set these values by editing the VAR file itself and then issuing a [reset] so that the EMC reads the new values. For our example let's directly edit the file so that G55 takes on the following values.

```

5241  2.000000
5242  1.000000
5243 -2.000000
5244  0.000000
5245  0.000000
5246  0.000000

```

You should read this as moving the zero positions of G55 to X = 2 units, Y= 1 unit, and Z = -2 units away from the absolute zero position.

Once there are values assigned, a call to G55 in a program block would shift the zero reference by the values stored. The following line would then move each axis to the new zero position. Unlike G53, G54 through G59.3 are modal commands. They will act on all blocks of code after one of them has been set. The program that might be run using figure 16.1 would require only a single coordinate reference for each of the locations and all of the work to be done there. The following code is offered as an example of making a square using the G55 offsets that we set above.

```

G55 G0 x0 y0 z0
g1 f2 z-0.2000
x1
y1

```

```
x0  
y0  
g0 z0  
g54 x0 y0 z0  
m2
```

“But,” you say, “why is there a G54 in there near the end.” Many programmers leave the G54 coordinate system with all zero values so that there is a modal code for the absolute machine based axis positions. This program assumes that we have done that and use the ending command as a command to machine zero. It would have been possible to use g53 and arrive at the same place but that command would not have been modal and any commands issued after it would have returned to using the G55 offsets because that coordinate system would still be in effect.

```
G54   use preset work coordinate system 1  
G55   use preset work coordinate system 2  
G56   use preset work coordinate system 3  
G57   use preset work coordinate system 4  
G58   use preset work coordinate system 5  
G59   use preset work coordinate system 6  
G59.1 use preset work coordinate system 7  
G59.2 use preset work coordinate system 8  
G59.3 use preset work coordinate system 9
```

### 16.3.1 Default coordinate system

One other variable in the VAR file becomes important when we think about offset systems. This variable is named 5220. In the default files it's value is set to 1.00000. This means that when the EMC starts up it should use the first coordinate system as its default. If you set this to 9.00000 it would use the ninth offset system as its default for startup and reset. Any value other than an interger (decimal really) between 1 and 9 will cause the EMC to fault on startup.

### 16.3.2 Setting coordinate system values within G-code.

In the general programming chapter we listed a G10 command word. This command can be used to change the values of the offsets in a coordinate system. (add here)

## 16.4 G92 Offsets

G92 is the most misunderstood and maligned part of EMC programming. The way that it works has changed just a bit from the early days to the current releases. This change has confused many users. It should be thought of as a temporary offset that is applied to all other offsets.

In response to criticism of it, Ray Henry studied it by comparing the way the interpreter authors expected it to work and the way that it worked on his Grizzly minimill. The following quoted paragraphs are extracted from his paper which is available in several text formats in the dropbox at <http://www.linuxcnc.org>.

### 16.4.1 The G92 commands

This set of commands include;

**G92** This command, when used with axis names, sets values to offset variables.

**G92.1** This command sets zero values to the g92 variables.

**G92.2** This command suspends but does not zero out the g92 variables.

**G92.3** This command applies offset values that have been suspended.

When the commands are used as described above, they will work pretty much as you would expect.

A user must understand the correct ways that the g92 values work. They are set based upon the location of each axis when the g92 command is invoked. The NIST document is clear that, "To make the current point have the coordinates" x0, y0, and z0 you would use g92 x0 y0 z0. *G92 does not work from absolute machine coordinates. It works from current location.*

G92 also works from current location as modified by any other offsets that are in effect when the g92 command is invoked. While testing for differences between work offsets and actual offsets it was found that a g54 offset could cancel out a g92 and thus give the appearance that no offsets were in effect. However, the g92 was still in effect for all coordinates and did produce expected work offsets for the other coordinate systems.

It is likely that the absence of home switches and proper home procedures will result in very large errors in the application of g92 values if they exist in the var file. Many EMC users do not have home switches in place on their machines. For them home should be found by moving each axis to a location and issuing the home command. When each axis is in a known location, the home command will recalculate how the g92 values are applied and will produce consistent results. Without a home sequence, the values are applied to the position of the machine when the EMC begins to run.

### 16.4.2 Setting G92 values

There are at least two ways to set G92 values.

- right mouse click on position displays of tkemc will popup a window into which you can type a value.
- the g92 command

Both of these work from the current location of the axis to which the offset is to be applied.

Issuing g92 x y z a b c does in fact set values to the g92 variables such that each axis takes on the value associated with it's name. These values are assigned to the current position of the machine axis. These results satisfy paragraphs one and two of the NIST document.

G92 commands work from current axis location and add and subtract correctly to give the current axis position the value assigned by the g92 command. The effects work even though previous offsets are in.

So if the X axis is currently showing 2.0000 as its position a G92 x0 will set an offset of -2.0000 so that the current location of X becomes zero. A G92 X2 will set an offset of 0.0000 and the displayed position will not change. A G92 X5.0000 will set an offset of 3.0000 so that the current displayed position becomes 5.0000.

### 16.4.3 G92 Cautions

Sometimes the values of a G92 offset get stuck in the VAR file. When this happens reset or a startup will cause them to become active again. The variables are named

```
5211  0.000000
5212  0.000000
5213  0.000000
5214  0.000000
5215  0.000000
5216  0.000000
```

where 5211 is the X axis offset and so on. If you are seeing unexpected positions as the result of a commanded move, or even unexpected numbers in the position displays when you start up, look at these variables in the VAR file and see if they contain values. If they do, set them to zeros and the problems should go away.

With these tests we can see that reset returns g92 to the condition that it had when the interpreter started up. The reader should note that we have established ... that no write of these values occurs during a normal run so if no g92 was set at the startup, none will be read in during a reset.

It may be that this is the heart of the problem that some have experienced with differences between the old and the new interpreter. It may well be, but I leave it to others to test, that the old interpreter and task programs immediately wrote values to the var file and then found those values during a reset.

On the other hand, if G92 values existed in the VAR file when the EMC started up

... starting the EMC with g92 values in the var file is that it will apply the values to current location of each axis. If this is home position and home position is set as machine zero everything will be correct. Once home has been established using real machine switches or moving each axis to a known home position and issuing an axis home command, g92 commands and values work as advertised.

These tests did not study the effect of re-reading the var file while they contain numbers. This could cause problems if g92 offsets had been removed with g92.1 but the var file still contained the previous numbers.

It is this complexity that causes us to say that G92 values must be treated as temporary. They should be used to set global short term offsets. The G54-59.3 coordinate systems should be used whenever long lasting and predictable offsets are needed.

## 16.5 Sample Program Using Offsets

This sample engraving project mills a set of four .1 radius circles in roughly a star shape around a center circle. We can setup the individual circle pattern like this.

```
G10 L2 P1 x0 y0 z0 (ensure that g54 is set to machine zero)
g0 x-.1 y0 z0
g1 f1 z-.25
g3 x-.1 y0 i.1 j0
g0 z0
m2
```

We can issue a set of commands to create offsets for the four other circles like this.

```
G10 L2 P2 x0.5 (offsets g55 x value by 0.5 inch)
G10 L2 P3 x-0.5 (offsets g56 x value by -0.5 inch)
G10 L2 P4 y0.5 (offsets g57 y value by 0.5 inch)
G10 L2 P5 y-0.5 (offsets g58 y value by -0.5 inch)
```

We put these together in the following program.

(a program for milling five small circles in a diamond shape)

```
G10 L2 P1 x0 y0 z0 (ensure that g54 is machine zero)
G10 L2 P2 x0.5 (offsets g55 x value by 0.5 inch)
G10 L2 P3 x-0.5 (offsets g56 x value by -0.5 inch)
G10 L2 P4 y0.5 (offsets g57 y value by 0.5 inch)
G10 L2 P5 y-0.5 (offsets g58 y value by -0.5 inch)
g54 g0 x-.1 y0 z0 (center circle)
g1 f1 z-.25
g3 x-.1 y0 i.1 j0
g0 z0
g55 g0 x-.1 y0 z0 (first offset circle)
g1 f1 z-.25
g3 x-.1 y0 i.1 j0
g0 z0
g56 g0 x-.1 y0 z0 (second offset circle)
g1 f1 z-.25
g3 x-.1 y0 i.1 j0
g0 z0
g57 g0 x-.1 y0 z0 (third offset circle)
g1 f1 z-.25
g3 x-.1 y0 i.1 j0
g0 z0
g58 g0 x-.1 y0 z0 (fourth offset circle)
g1 f1 z-.25
g3 x-.1 y0 i.1 j0
g54 g0 x0 y0 z0
m2
```

Now comes the time when we might apply a set of G92 offsets to this program. You'll see that it is running in each case at z0. If the mill were at the zero position, a g92 z1.0000 issued at the head of the program would shift everything down an inch. You might also shift the whole pattern around in the XY plane by adding some x and y offsets with g92. If you do this you should add a G92.1 command just before the m2 that ends the program. If you do not, other programs that you might run after this one will also use that g92 offset. Furthermore it would save the g92 values when you shut down the EMC and they will be recalled when you start up again.

## Appendix A

# Glossary of Common Terms Used in the EMC Documents

GPLD Copyright 2003, LinuxCNC.org

A listing of terms and what they mean. Some terms have a general meaning and several additional meanings for users, installers, and developers.

**Acme Screw** A type of lead-screw [A](#) that uses an acme thread form. Acme threads have somewhat lower friction and wear than simple triangular threads, but ball-screws [A](#) are lower yet. Most manual machine tools use acme lead-screws.

**Axis** One of the computer control movable parts of the machine. For a typical vertical mill, the table is the X axis, the saddle is the Y axis, and the quill or knee is the Z axis. Additional linear axes parallel to X, Y, and Z are called U, V, and W respectively. Angular axes like rotary tables are referred to as A, B, and C.

**Backlash** The amount of "play" or lost motion that occurs when direction is reversed in a lead screw [A](#). or other mechanical motion driving system. It can result from nuts that are loose on leadscrews, slippage in belts, cable slack, "wind-up" in rotary couplings, and other places where the mechanical system is not "tight". Backlash will result in inaccurate motion, or in the case of motion caused by external forces (think cutting tool pulling on the work piece) the result can be broken cutting tools. This can happen because of the sudden increase in chip load on the cutter as the work piece is pulled across the backlash distance by the cutting tool.

**Backlash Compensation** - Any technique that attempts to reduce the effect of backlash without actually removing it from the mechanical system. This is typically done in software in the controller. This can correct the final resting place of the part in motion but fails to solve problems related to direction changes while in motion (think circular interpolation) and motion that is caused when external forces (think cutting tool pulling on the work piece) are the source of the motion.

**Ball Screw** A type of lead-screw that uses small hardened steel balls between the nut [A](#) and screw to reduce friction. Ball-screws have very low friction and backlash [A](#), but are usually quite expensive.

**Ball Nut** A special nut designed for use with a ball-screw. It contains an internal passage to re-circulate the balls from one end of the screw to the other.

**bridgeportio** An I/O task [A](#) designed to work with a Bridgeport milling machine, having a variable speed spindle [A](#), coolant, and lube pumps, and some other stuff.

**CNC** Computer Numerical Control. The general term used to refer to computer control of machinery. Instead of a human operator turning cranks to move a cutting tool, CNC uses a computer and motors to move the tool, based on a part program [A](#).



**Coordinate Measuring Machine** A Coordinate Measuring Machine is used to make many accurate measurements on parts. These machines can be used to create CAD data for parts where no drawings can be found, when a hand-made prototype needs to be digitized for moldmaking, or to check the accuracy of machined or molded parts.

**DRO** A Digital Read Out is a device attached to the slides of a machine tool or other device which has parts that move in a precise manner to indicate the current location of the tool with respect to some reference position. Nearly all DRO's use linear quadrature encoders to pick up position information from the machine.

**EDM** EDM is a method of removing metal in hard or difficult to machine or tough metals, or where rotating tools would not be able to produce the desired shape in a cost-effective manner. An excellent example is rectangular punch dies, where sharp internal corners are desired. Milling operations can not give sharp internal corners with finite diameter tools. A wire EDM machine can make internal corners with a radius only slightly larger than the wire's radius. A 'sinker' EDM can make corners with a radius only slightly larger than the radius on the corner of the convex EDM electrode.

**EMC** The Enhanced Machine Controller. Initially a NIST [A](#) project. EMC is able to run a wide range of motion devices.

**EMCIO** The module within [EMCA](#) that handles general purpose I/O, unrelated to the actual motion of the axes. A couple examples are [bridgeportio A](#) and [minimillio A](#).

**EMCMOT** The module within [EMC A](#) that handles the actual motion of the cutting tool. It runs as a real-time program and directly controls the motors.

### **Encoder**

**Feed** Relatively slow, controlled motion of the tool used when making a cut.

**Feedrate** The speed at which a motion occurs. In manual mode, jog speed can be set from the graphical interface. In auto or mdi mode feedrate is commanded using a (f) word. F10 would mean ten units per minute.

### **Feedback**

**Feedrate Override** A manual, operator controlled change in the rate at which the tool moves while cutting. Often used to allow the operator to adjust for tools that are a little dull, or anything else that requires the feed rate to be "tweaked".

**G-Code** The generic term used to refer to the most common part programming language. There are several dialects of G-code, EMC uses RS274/NGC [A](#).

**GUI** Graphical User Interface.

**General** A type of interface that allows communications between a computer and human (in most cases) via the manipulation of icons and other elements (widgets) on a computer screen.

**EMC** An application that presents a graphical screen to the machine operator allowing manipulation of machine and the corresponding controlling program.

**Home** A specific location in the machine's work envelope that is used to make sure the computer and the actual machine both agree on the tool position.

**ini file** A text file that contains most of the information that configures [EMC A](#) for a particular machine

**Joint\_Coordinates:** These specify the angles between the individual joints of the machine. Kinematics

**Jog** Manually moving an axis of a machine. Jogging either moves the axis a fixed amount for each key-press, or moves the axis at a constant speed as long as you hold down the key.

### kernel-space

**Kinematics** The position relationship between world coordinates [A](#) and joint coordinates [A](#) of a machine. There are two types of kinematics. Forward kinematics is used to calculate world coordinates from joint coordinates. Inverse kinematics is used for exactly opposite purpose. Note that kinematics does not take into account, the forces, moments etc. on the machine. It is for positioning only.

**Lead-screw** An screw that is rotated by a motor to move a table or other part of a machine. Lead-screws are usually either ball-screws [A](#) or acme screws [A](#), although conventional triangular threaded screws may be used where accuracy and long life are not as important as low cost.

**MDI** Manual Data Input. This is a mode of operation where the controller executes single lines of G-code [A](#) as they are typed by the operator.

**minimillio** An I/O task [A](#) designed to work with small table-top mills.

**NIST** National Institute of Standards and Technology. An agency of the Department of Commerce in the United States.

### Offsets

**Part Program** A description of a part, in a language that the controller can understand. For EMC, that language is RS-274/NGC, commonly known as G-code [A](#).

**Rapid** Fast, possibly less precise motion of the tool, commonly used to move between cuts. If the tool meets the material during a rapid, it is probably a bad thing!

**Real-time** Software that is intended to meet very strict timing deadlines. Under Linux, in order to meet these requirements it is necessary to install RTAI [A](#) or RTLINUX [A](#) and build the software to run in those special environments. For this reason real-time software runs in kernel-space.

**RTAI** Real Time Application Interface, see <http://www.aero.polimi.it/~rtai/> <http://www.aero.polimi.it/~rtai/> one of two real-time extensions for Linux that EMC can use to achieve real-time [A](#) performance.

**RTLINUX** See <http://www.rtlinux.org> <http://www.rtlinux.org>, one of two real-time extensions for Linux that EMC can use to achieve real-time [A](#) performance.

**RS-274/NGC** The formal name for the language used by EMC [A](#) part programs [A](#).

### Servo Motor

#### Servo Loop

**Spindle** On a mill or drill, the spindle holds the cutting tool. On a lathe, the spindle holds the workpiece.

**Stepper Motor** A type of motor that turns in fixed steps. By counting steps, it is possible to determine how far the motor has turned. If the load exceeds the torque capability of the motor, it will skip one or more steps, causing position errors.

**TASK** The module within EMC [A](#) that coordinates the overall execution and interprets the part program.

**Tcl/Tk** A scripting language and graphical widget toolkit with which EMC's most popular GUI's [A](#) were written.

**World Coordinates:** This is the absolute frame of reference. It gives coordinates in terms of a fixed reference frame that is attached to some point (generally the base) of the machine tool.

# Appendix A

## Legal Section

### Handbook Copyright Terms

Copyright (c) 2000 LinuxCNC.org

Permission is granted to copy, distribute and/or modify this document under the terms of the GNU Free Documentation License, Version 1.1 or any later version published by the Free Software Foundation; with no Invariant Sections, no Front-Cover Texts, and one Back-Cover Text: "This EMC Handbook is the product of several authors writing for linuxCNC.org. As you find it to be of value in your work, we invite you to contribute to its revision and growth." A copy of the license is included in the section entitled "GNU Free Documentation License". If you do not find the license you may order a copy from Free Software Foundation, Inc. 59 Temple Place, Suite 330, Boston, MA 02111-1307

#### A.1 GNU Free Documentation License Version 1.1, March 2000

Copyright (C) 2000 Free Software Foundation, Inc. 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

##### A.1.1 GNU Free Documentation License Version 1.1, March 2000

Copyright (C) 2000 Free Software Foundation, Inc. 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

###### 0. PREAMBLE

The purpose of this License is to make a manual, textbook, or other written document "free" in the sense of freedom: to assure everyone the effective freedom to copy and redistribute it, with or without modifying it, either commercially or noncommercially. Secondly, this License preserves for the author and publisher a way to get credit for their work, while not being considered responsible for modifications made by others.

This License is a kind of "copyleft", which means that derivative works of the document must themselves be free in the same sense. It complements the GNU General Public License, which is a copyleft license designed for free software.

We have designed this License in order to use it for manuals for free software, because free software needs free documentation: a free program should come with manuals providing the same freedoms that the software does. But this License is not limited to software manuals; it can be used for any textual work, regardless of subject matter or whether it is published as a printed book. We recommend this License principally for works whose purpose is instruction or reference.

###### 1. APPLICABILITY AND DEFINITIONS

This License applies to any manual or other work that contains a notice placed by the copyright holder saying it can be distributed under the terms of this License. The "Document", below, refers to any such manual or work. Any member of the public is a licensee, and is addressed as "you".

A "Modified Version" of the Document means any work containing the Document or a portion of it, either copied verbatim, or with modifications and/or translated into another language.

A "Secondary Section" is a named appendix or a front-matter section of the Document that deals exclusively with the relationship of the publishers or authors of the Document to the Document's overall subject (or to related matters) and contains nothing that could fall directly within that overall subject. (For example, if the Document is in part a textbook of mathematics, a Secondary Section may not explain any mathematics.) The relationship could be a matter of historical connection with the subject or with related matters, or of legal, commercial, philosophical, ethical or political position regarding them.

The "Invariant Sections" are certain Secondary Sections whose titles are designated, as being those of Invariant Sections, in the notice that says that the Document is released under this License.

The "Cover Texts" are certain short passages of text that are listed, as Front-Cover Texts or Back-Cover Texts, in the notice that says that the Document is released under this License.

A "Transparent" copy of the Document means a machine-readable copy, represented in a format whose specification is available to the general public, whose contents can be viewed and edited directly and straightforwardly with generic text editors or (for images composed of pixels) generic paint programs or (for drawings) some widely available drawing editor, and that is suitable for input to text formatters or for automatic translation to a variety of formats suitable for input to text formatters. A copy made in an otherwise Transparent file format whose markup has been designed to thwart or discourage subsequent modification by readers is not Transparent. A copy that is not "Transparent" is called "Opaque".

Examples of suitable formats for Transparent copies include plain ASCII without markup, Texinfo input format,  $\text{\LaTeX}$  input format, SGML or XML using a publicly available DTD, and standard-conforming simple HTML designed for human modification. Opaque formats include PostScript, PDF, proprietary formats that can be read and edited only by proprietary word processors, SGML or XML for which the DTD and/or processing tools are not generally available, and the machine-generated HTML produced by some word processors for output purposes only.

The "Title Page" means, for a printed book, the title page itself, plus such following pages as are needed to hold, legibly, the material this License requires to appear in the title page. For works in formats which do not have any title page as such, "Title Page" means the text near the most prominent appearance of the work's title, preceding the beginning of the body of the text.

## 2. VERBATIM COPYING

You may copy and distribute the Document in any medium, either commercially or noncommercially, provided that this License, the copyright notices, and the license notice saying this License applies to the Document are reproduced in all copies, and that you add no other conditions whatsoever to those of this License. You may not use technical measures to obstruct or control the reading or further copying of the copies you make or distribute. However, you may accept compensation in exchange for copies. If you distribute a large enough number of copies you must also follow the conditions in section 3.

You may also lend copies, under the same conditions stated above, and you may publicly display copies.

## 3. COPYING IN QUANTITY

If you publish printed copies of the Document numbering more than 100, and the Document's license notice requires Cover Texts, you must enclose the copies in covers that carry, clearly and legibly, all these Cover Texts: Front-Cover Texts on the front cover, and Back-Cover Texts on the back cover. Both covers must also clearly and legibly identify you as the publisher of these copies. The front cover must present the full title with all words of the title equally prominent and visible. You may add other material on the covers in addition.

Copying with changes limited to the covers, as long as they preserve the title of the Document and satisfy these conditions, can be treated as verbatim copying in other respects.

If the required texts for either cover are too voluminous to fit legibly, you should put the first ones listed (as many as fit reasonably) on the actual cover, and continue the rest onto adjacent pages.

If you publish or distribute Opaque copies of the Document numbering more than 100, you must either include a machine-readable Transparent copy along with each Opaque copy, or state in or with each Opaque copy a publicly-accessible computer-network location containing a complete Transparent copy of the Document, free of added material, which the general network-using public has access to download anonymously at no charge using public-standard network protocols. If you use the latter option, you must take reasonably prudent steps, when you begin distribution of Opaque copies in quantity, to ensure that this Transparent copy will remain thus accessible at the stated location until at least one year after the last time you distribute an Opaque copy (directly or through your agents or retailers) of that edition to the public.

It is requested, but not required, that you contact the authors of the Document well before redistributing any large number of copies, to give them a chance to provide you with an updated version of the Document.

## 4. MODIFICATIONS

You may copy and distribute a Modified Version of the Document under the conditions of sections 2 and 3 above, provided that you release the Modified Version under precisely this License, with the Modified Version filling the role of the Document, thus licensing distribution and modification of the Modified Version to whoever possesses a copy of it. In addition, you must do these things in the Modified Version:

A. Use in the Title Page (and on the covers, if any) a title distinct from that of the Document, and from those of previous versions (which should, if there were any, be listed in the History section of the Document). You may use the same title as a previous version if the original publisher of that version gives permission. B. List on the Title Page, as authors, one or more persons or entities responsible for authorship of the modifications in the Modified Version, together with at least five of the principal authors of the Document (all of its principal authors, if it has less than five). C. State on the Title page the name of the publisher of the Modified Version, as the publisher. D. Preserve all the copyright notices of the Document. E. Add an appropriate copyright notice for your modifications adjacent to the other copyright notices. F. Include, immediately after the copyright notices, a license notice giving the public permission to use the Modified Version under the terms of this License, in the form shown in the Addendum below. G. Preserve in that license notice the full lists of Invariant Sections and required Cover Texts given in the Document's license notice. H. Include an unaltered copy of this License. I. Preserve the section entitled "History", and its title, and add to it an item stating at least the title, year, new authors, and publisher of the Modified Version as given on the Title Page. If there is no section entitled "History" in the Document, create one stating the title, year, authors, and publisher of the Document as given on its Title Page, then add an item describing the Modified Version as stated in the previous sentence. J. Preserve the network location, if any, given in the Document for public access to a Transparent copy of the Document, and likewise the network locations given in the Document for previous versions it was based on. These may be placed in the "History" section. You may omit a network location for a work that was published at least four years before the Document itself, or if the original publisher of the version it refers to gives permission. K. In any section entitled "Acknowledgements" or "Dedications", preserve the section's title, and preserve in the section all the substance and tone of each of the contributor acknowledgements and/or dedications given therein. L. Preserve all the Invariant Sections of the Document, unaltered in their text and in their titles. Section numbers or the equivalent are not considered part of the section titles. M. Delete any section entitled "Endorsements". Such a section may not be included in the Modified Version. N. Do not retitle any existing section as "Endorsements" or to conflict in title with any Invariant Section.

If the Modified Version includes new front-matter sections or appendices that qualify as Secondary Sections and contain no material copied from the Document, you may at your option designate some or all of these sections as invariant. To do this, add their titles to the list of Invariant Sections in the Modified Version's license notice. These titles must be distinct from any other section titles.

You may add a section entitled "Endorsements", provided it contains nothing but endorsements of your Modified Version by various parties—for example, statements of peer review or that the text has been approved by an organization as the authoritative definition of a standard.

You may add a passage of up to five words as a Front-Cover Text, and a passage of up to 25 words as a Back-Cover Text, to the end of the list of Cover Texts in the Modified Version. Only one passage of Front-Cover Text and one of Back-Cover Text may be added by (or through arrangements made by) any one entity. If the Document already includes a cover text for the same cover, previously added by you or by arrangement made by the same entity you are acting on behalf of, you may not add another; but you may replace the old one, on explicit permission from the previous publisher that added the old one.

The author(s) and publisher(s) of the Document do not by this License give permission to use their names for publicity for or to assert or imply endorsement of any Modified Version.

## 5. COMBINING DOCUMENTS

You may combine the Document with other documents released under this License, under the terms defined in section 4 above for modified versions, provided that you include in the combination all of the Invariant Sections of all of the original documents, unmodified, and list them all as Invariant Sections of your combined work in its license notice.

The combined work need only contain one copy of this License, and multiple identical Invariant Sections may be replaced with a single copy. If there are multiple Invariant Sections with the same name but different contents, make the title of each such section unique by adding at the end of it, in parentheses, the name of the original author or publisher of that section if known, or else a unique number. Make the same adjustment to the section titles in the list of Invariant Sections in the license notice of the combined work.

In the combination, you must combine any sections entitled "History" in the various original documents, forming one section entitled "History"; likewise combine any sections entitled "Acknowledgements", and any sections entitled "Dedications". You must delete all sections entitled "Endorsements."

## 6. COLLECTIONS OF DOCUMENTS

You may make a collection consisting of the Document and other documents released under this License, and replace the individual copies of this License in the various documents with a single copy that is included in the collection, provided that you follow the rules of this License for verbatim copying of each of the documents in all other respects.

You may extract a single document from such a collection, and distribute it individually under this License, provided you insert a copy of this License into the extracted document, and follow this License in all other respects regarding verbatim copying of that document.

### **7. AGGREGATION WITH INDEPENDENT WORKS**

A compilation of the Document or its derivatives with other separate and independent documents or works, in or on a volume of a storage or distribution medium, does not as a whole count as a Modified Version of the Document, provided no compilation copyright is claimed for the compilation. Such a compilation is called an "aggregate", and this License does not apply to the other self-contained works thus compiled with the Document, on account of their being thus compiled, if they are not themselves derivative works of the Document.

If the Cover Text requirement of section 3 is applicable to these copies of the Document, then if the Document is less than one quarter of the entire aggregate, the Document's Cover Texts may be placed on covers that surround only the Document within the aggregate. Otherwise they must appear on covers around the whole aggregate.

### **8. TRANSLATION**

Translation is considered a kind of modification, so you may distribute translations of the Document under the terms of section 4. Replacing Invariant Sections with translations requires special permission from their copyright holders, but you may include translations of some or all Invariant Sections in addition to the original versions of these Invariant Sections. You may include a translation of this License provided that you also include the original English version of this License. In case of a disagreement between the translation and the original English version of this License, the original English version will prevail.

### **9. TERMINATION**

You may not copy, modify, sublicense, or distribute the Document except as expressly provided for under this License. Any other attempt to copy, modify, sublicense or distribute the Document is void, and will automatically terminate your rights under this License. However, parties who have received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.

### **10. FUTURE REVISIONS OF THIS LICENSE**

The Free Software Foundation may publish new, revised versions of the GNU Free Documentation License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns. See <http://www.gnu.org/copyleft/>.

Each version of the License is given a distinguishing version number. If the Document specifies that a particular numbered version of this License "or any later version" applies to it, you have the option of following the terms and conditions either of that specified version or of any later version that has been published (not as a draft) by the Free Software Foundation. If the Document does not specify a version number of this License, you may choose any version ever published (not as a draft) by the Free Software Foundation.

#### **ADDENDUM:** How to use this License for your documents

To use this License in a document you have written, include a copy of the License in the document and put the following copyright and license notices just after the title page:

Copyright (c) YEAR YOUR NAME. Permission is granted to copy, distribute and/or modify this document under the terms of the GNU Free Documentation License, Version 1.1 or any later version published by the Free Software Foundation; with the Invariant Sections being LIST THEIR TITLES, with the Front-Cover Texts being LIST, and with the Back-Cover Texts being LIST. A copy of the license is included in the section entitled "GNU Free Documentation License".

If you have no Invariant Sections, write "with no Invariant Sections" instead of saying which ones are invariant. If you have no Front-Cover Texts, write "no Front-Cover Texts" instead of "Front-Cover Texts being LIST"; likewise for Back-Cover Texts.

If your document contains nontrivial examples of program code, we recommend releasing these examples in parallel under your choice of free software license, such as the GNU General Public License, to permit their use in free software.

# Index

G-code, [14](#)  
G53, [113](#)  
G54, [115](#)  
G55, [113](#), [115](#)  
G56, [115](#)  
G57, [115](#)  
G58, [115](#)  
G59, [115](#)