SECTION 7 OPERATION

BAAM CI

A SUPPLEMENT TO THE OPERATION MANUAL FOR THE

BAAM 100

CINCINNATI®

CINCINNATI INCORPORATED
CINCINNATI, OHIO 45211

EM-572 (N-12-16)

SECTION 7: OPERATION CONTENTS

INTRODUCTION

HMI OVERVIEW

MENU COMMANDS

VIEW MENU

MAINTENANCE MENU

RUNNING A PROGRAM

RESTARTING A PROGRAM - ERROR RECOVERY

ALARMS AND MESSAGES

NC PROGRAMMING

CINCINNATI BAAM SYSTEM SECTION 7

CINCINNATI BAAM 100 SERIES LASER SYSTEM

Supplement Manual, Section 7 - OPERATION, for BAAM 100

This manual applies to all BAAM Series System

HMI OVERVIEW

The Human Machine Interface (HMI) is the means provided for interacting with the BAAM control. This chapter presents an overview of the software user interface. The foundation of the software user interface is the *BAAM System Display* window.

HMI START-UP AND SHUT-DOWN

HMI START-UP

1. To operate the system, log on to the Microsoft Windows® - based PC control to start the system **CINCINNATI BAAM HMI** application. If the previous user did not log off, then the **CINCINNATI BAAM HMI** program should be running. If the screen saver is displayed on the control station monitor, then move the trackball or press any keyboard key to restore the display. If the CNC/HMI user interface is not visible, then proceed with the following instructions to log on.

2. When the control is first powered-up, the Windows Logon screen will be displayed on the control station monitor. To log on, click a user name and enter the corresponding password. There are different user logon levels, starting with the Operator user.

USER LOGON LEVELS

Operator: For normal machine operation, select "Operator" for the user name and enter "driver" for the password (without quotation marks). The System **CINCINNATI BAAM HMI** application will start automatically after each Operator logon. The Operator user has access to all user interface controls needed to prepare the System for operation, load and execute programs. Operator has limited access to *Machine Configuration* settings.

Note: *The user name is NOT letter case sensitive. Any combination of upper or lowercase letters can be entered for the user name. The password* ***is*** *case sensitive. The user must enter the appropriate password exactly as instructed.*

Manager: The Manager level extends Operator-level access by adding the capability to perform administrative functions such as installing software updates, setting the system date/time, backing-up/restoring files and configuring the automatic backup system. See the Windows Administration topic for more about these administrative functions. The Manager logon level also provides edit access to all *Machine Configuration* parameters.

Setup: The Setup level combines the administrative capabilities of the Manager level with network/domain setup and configuration capabilities. The Setup level is limited to performing administrative functions. The system can not operate using the Setup logon level.

3. If the **BAAM HMI** program is not already running, start it now by clicking the '**CINCINNATI BAAM HMI**' icon.

SHUTTING DOWN THE HMI

1. When shutting down the HMI application, the material that is loaded in the receiver hopper should be purged in the extruder.

2. Turn off the extruder zone through the heating software.

3. Turn off the table zone temperature through the heating software. To prevent the warping of the build sheet it is recommended to leave the vacuum on until the table temperature is room temperature.

4. Turn off the vacuum. To turn off the vacuum, press the button when green, a message window will pop up asking if you

want to turn off the vacuum. Press OK to turn off the vacuum, this prevents accidently turning off the vacuum. If the vacuum is used, it is recommended to leave the vacuum on the entire print time and cooling time to prevent warping of the build sheets or other printed part.

5. Switch off the main drives by rotating the "DRIVES" keyswitch on the Control Station Side Panel to the "LOCK/OFF" position.

6. With the main drives off, the chiller will shut down automatically if the chiller switch is in the "REMOTE" position. If the chiller is in Local mode, switch it to "STANDBY".

7. Terminate the CNC/HMI application using any of the application **Exit** commands.

   Note: CINCINNATI INCORPORATED recommends leaving the machine disconnect on at all times unless maintenance or repair procedures require turning off the power to the machine. If no such maintenance or repairs are required, the above steps complete a normal shutdown. If machine disconnect must be turned off, proceed to the next step.

8. If the main power disconnect is turned off, be sure to wait for the Operator Control Station screen to go completely blank before servicing the System computer. If equipped, the computer is powered by an uninterruptible power supply (UPS), power to the computer will remain on for several minutes after the disconnect is turned off.. After the main power disconnect is turned off, the operating system will begin the automatic shutdown procedure. When automatic shutdown is complete, the Operator Control Station screen will be blank, indicating the UPS has turned off power to the computer.

9. The system computer has a UPS (Uninterruptible Power Supply) for power protection. After main power shutdown, the power to the computer will remain on for up to 10 minutes before shutting down.

10. To prevent unauthorized machine operation, remove keys from all key-operated control switches.

    Note: Some keys can operate more than one switch.

SYSTEM DISPLAY WINDOW

The *System Display* window occupies the entire display screen on the operator control station. The main components of the *System Display* window are the Title Bar, Menu Bar, Tool Bar, Status Indicators, Application Workspace and Operator Console Control Bar.

MANUAL MODE DISPLAY

In Manual mode the manual indicator lights up and you can access the operating modes and Axes controls.

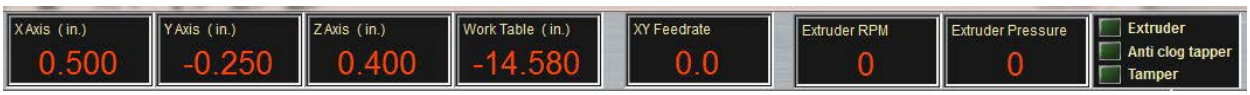
MENU BAR

The Menu Bar is the row of main menu icons across the bottom of the System Display window in both the Manual and Auto modes. Selecting a menu title opens a pop-up menu displaying a list of menu commands.

File Menu	View Menu	Mainten -ance Menu	Help / About Menu	Save	Config- uration	Open CNC File	Load CNC File	View Log Files	Time /Date Display

INFORMATION BAR

The Information Bar is the row of main menu icons across the top of the System Display window in both the Manual and Auto modes.



X Axis	*Y Axis*	*Z Axis*	*Work Table*	*XY Feedrate*	*Extruder RPM*	*Extruder pressure*	*Extruder Anti Clog Tapper Tamper*

AUTO MODE DISPLAY

STATUS INDICATORS CONTROL BAR

The left-most indicator shows the current alarm status. It will be Red if any system alarms are present and Yellow if any FYI messages are active. Clicking on this indicator will open the "System Alarms and Messages" window. The middle indicator shows the current Access Level. Clicking in this box will display the Change Access Level" window. The right most indicator shows the current Units Display. Clicking on this indicator will toggle between Inch mode and Metric mode.

FILE TYPES

The BAAM System control software gets the information it needs to execute the applications from NC Program files.

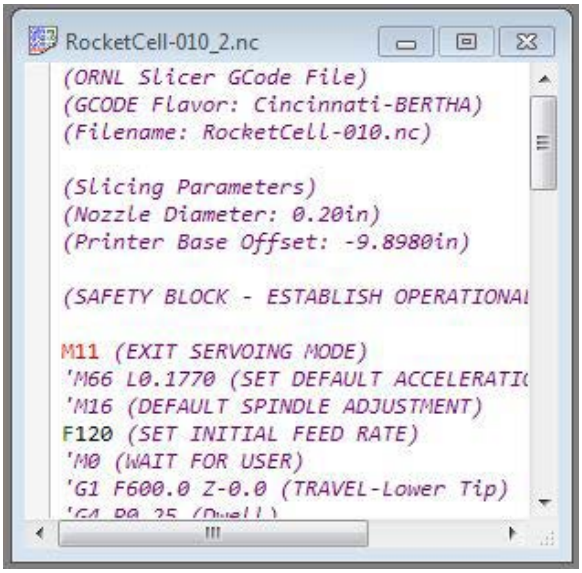
The *File* menu commands and Tool Bar buttons, described later, can be used to open these files for viewing/editing, and to create new files. When opening an existing file or creating a new file, a window will appear in the Application Workspace, displaying the file's contents. The Title Bar of each file view window contains the name of its file.

When a file is modified, it must be saved using **File | Save** or Tool Bar **Save** command before any changes will take effect. An asterisk "*****" will appear next to the file name of any file that has been modified but not saved.

NC PROGRAM FILE

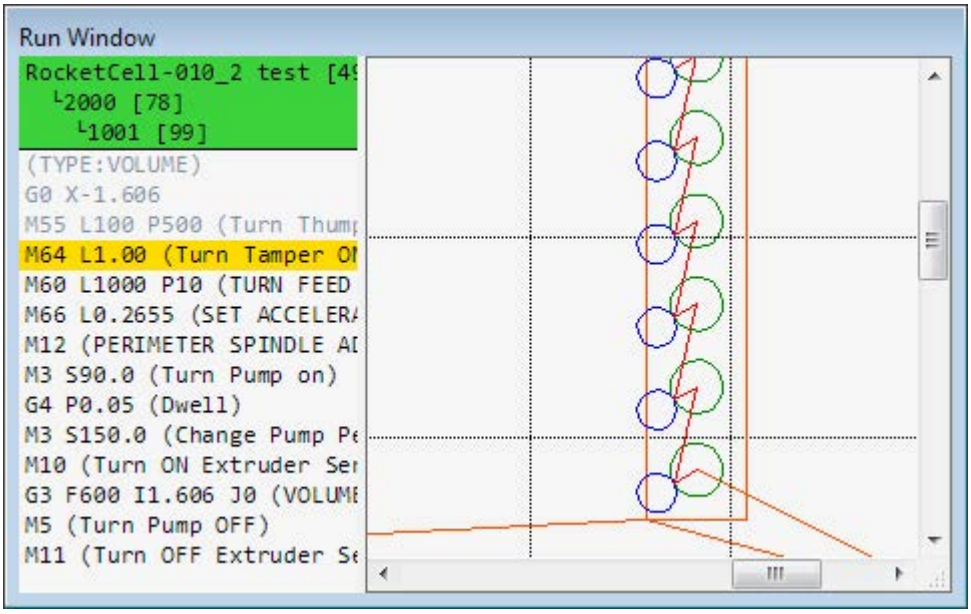
The NC Program file contains the NC codes that determine the part feature geometry and control the various machine printing functions.

When an NC Program file is opened, its contents are displayed in a *Program Edit* window. *Program Edit* windows use context coloring; distinct program components are displayed with different colors (G codes are blue, M codes are red, etc).



CNC RUN WINDOW

The *CNC Run Window* displays the execution status of the active program, if a program is currently loaded. This window cannot be manually opened or closed. The control software will automatically display the *CNC Run Window* each time a program is successfully loaded, and whenever the control enters Auto mode while a program is loaded. Likewise, the *CNC Run Window* will automatically close whenever the control enters Jog mode.



The *CNC Run Window* has two sections. The left section displays the NC program code with the active block highlighted. To help locate the current position in an interrupted program, the active subroutine name and line number are displayed in a green box at the top of the NC code section. All currently active subroutines are displayed, indented to the right to indicate the nesting depth for nested subroutines.

The right section is a graphical plot of the current layer in the program.

To change the width of the plot section, select the divider between the left and right sections and drag it to another location. To change the viewing area of the plot, use any of the various *Zoom* features. See **WINDOW ZOOMING** in the **VIEW MENU** section.

CONTROL BARS

PROGRAM MODES

The right side of the display window contains controls that turn on or off the various program execution modes that affect how the control software executes NC programs while in Auto mode. Each control is both a button and indicator for the corresponding mode.

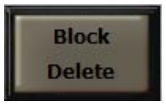
The BAAM Control Software automatically displays the Program Modes Control Bar each time the CNC control enters Auto mode. When the control enters Jog or Axes Home mode from Auto mode, this control bar will automatically close.

PROGRAM TEST MODE

Program Test Mode will run a part program without actually printing material. Use the “Program Test” button to activate Program Test Mode. In this mode, the extruder will *not* turn on during program execution.

SINGLE BLOCK MODE

In Single Block mode, only one NC Program block will be executed each time “CYCLE START” is pressed. The “Single Block” button activates Single Block program execution mode.



BLOCK DELETE MODE

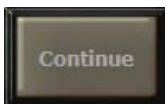
While Block Delete mode is active, any block in the NC program that begins with the “ / ” (forward slash) character will not be executed. The “Block Delete” button activates Block Delete mode. Block Delete mode can be enabled/disabled at any time during program execution.

OPTIONAL STOP MODE

Optional Stop mode enables the use of **M01** (or **M1**) in a program. Use the “Optional Stop” button to activate Optional Stop program execution mode. When Optional Stop mode is active, the control changes to the Cycle Stop condition when a program commands **M01**. Program execution resumes when “CYCLE START” is pressed. Optional Stop mode can be enabled/disabled at any time during program execution.

REWIND

The “Rewind” button sets the first block of the active NC program as the next block to be executed when “CYCLE START” is pressed. Use this function to restart the program from the beginning when program execution is interrupted. The “Rewind” button is disabled while program execution is in progress. Note that this button does not stay in the green (ON) state when selected, since it does not make a new mode active.

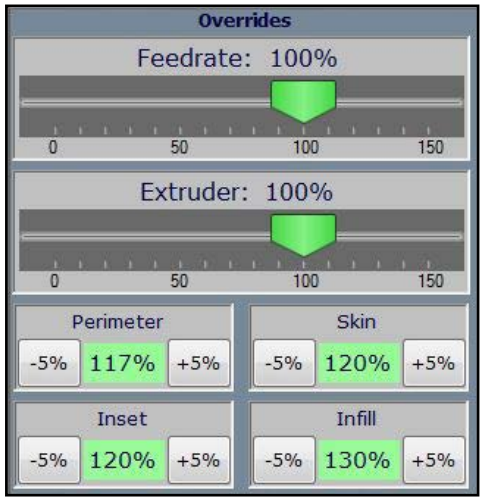


CONTINUE

The “Continue” button will flash when a G104 Dwell is active. Pushing it while it is flashing will abort the current dwell and continue with the current program execution.

OVERRIDE SETTINGS

On the left side of the display window, the current values of the feedrate and extruder override settings are shown.



The two large sliders control the overall feedrate and extruder overrides. Double clicking on either slider will quickly return the value to 100%. The 4 smaller indicators can be used to further control the extruder speed when the program is in the Perimeter, Skin, Inset, or Infill mode (see M12 – M16). These overrides are used in addition to the overall extruder override. For example, if the current program is in Perimeter mode (M12 active) and the Perimeter override is set at 115% and the overall Extruder override is set at 110%, the actual override in affect will be 1.15 * 1.10 = 126%. Double clicking on the override percent value will return the value to 100%. Holding down the Alt key on the keyboard will change the buttons from +/-5% increments to +/- 1% increments for finer control.

OPERATING MODES

The bottom section of the Operator Console Control Bar contains two button/indicator controls for the primary operating mode.

MANUAL

This button/indicator activates Manual or Jog mode. While the control is in Manual mode, the "Manual" indicator color is amber. In Manual mode, the axes motion controls on the Operator Console Control Bar can be used to manually move ("jog") each machine axis individually. The control will remain in Manual mode until another mode is selected.

AUTO

This button/indicator activates Auto mode. The System must be in Auto mode in order to run the programs. While the control is in Auto mode, the "Auto" indicator color is amber.

CYCLE START

This button/indicator starts the program execution in Auto mode and is used to initiate special functions in Manual mode.

CYCLE STOP

This button/indicator stops the program execution in Auto mode and stops any special functions in Manual mode.

RESET

This button/indicator is used to clear faults and warnings on the control.

SPECIAL OPERATING MODE

Special Operating Modes are alternate modes the BAAM System is in while certain functions are active:

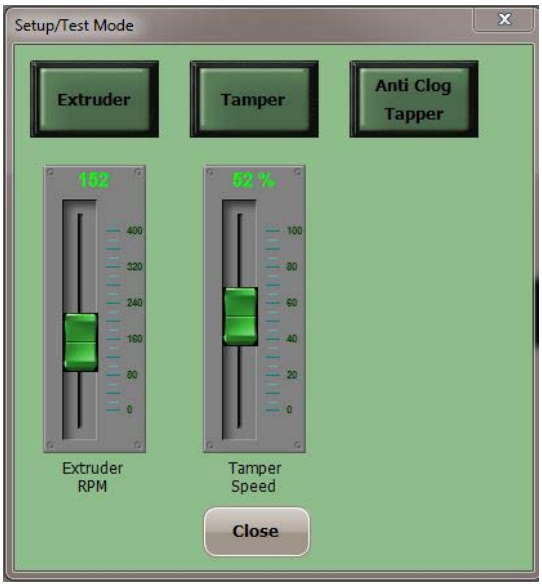
Note: *The BAAM System must be in Manual mode before a "Special Operating Mode" can be activated.*

HOME AXES

This button/indicator is used to home the axes when the control is first started. To home the axes, the main drives must be on with no system alarms present. The Axes Homing function begins when the operator presses the "CYCLE START" pushbutton. When all axes have moved to their reference positions, all axes motion stops, the Axes Home function ends and button will illuminate Green.

SETUP / TEST

This button/indicator will open a special dialog that can be used to turn the extruder, tamper, and anti-clog tapper on at various speeds. It can be used to test these systems or to purge material out of the extruder.



Press the Extruder, Tamper or Anti Clog Tapper buttons to turn these items on. The sliders can be used to adjust the speed of the Extruder and Tamper.

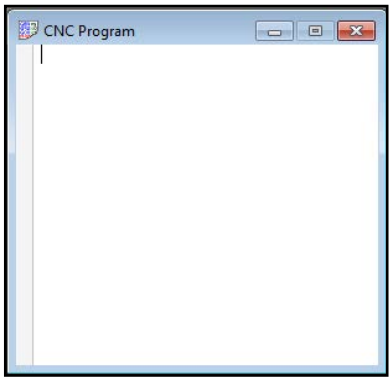
MENU COMMANDS

FILE MENU

In the *File* menu, various commands can be used to manage files and to activate file-related functions. Many of the *File* menu commands, such as **Open**, **Save**, and **Print**, are standard commands used by most Microsoft Windows applications, while others are for the BAAM System-specific functions: **Mid Program Start**, **Load Current File**, etc.

NEW

File | New Program File creates a new CNC file and opens a new window displaying the files contents.

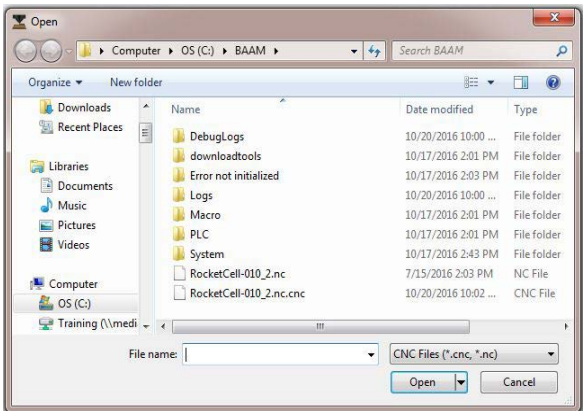


Keyboard Shortcut: ***Ctrl + N***

OPEN

File | Open Program File opens an existing file in a new window. Multiple file windows can be open at the same time. This command causes the *File Open* window to appear. Use this window to select a file to open.

Note: *Use the Most Recently Used (MRU) File List to quickly reopen a file that was recently closed. See the* ***MRU FILE LIST*** *section for more information.*



Keyboard Shortcut: ***Ctrl + O***

SAVE

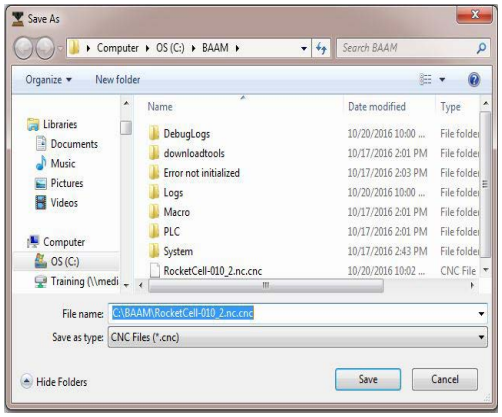
File | Save saves any changes to the active file. The contents of the file will be written to its current location with its current file name. When a new file is saved for the first time, the *Save As* window will open first, prompting the user to name the file.

Toolbar Shortcut: 

Keyboard Shortcut: ***Ctrl + S***

SAVE AS

File | Save As saves a new file with the specified name, or saves the contents of the active file to a different name and/or location. This command causes the *Save As* window to open. Use this window to specify the file name and location.



LOAD CURRENT FILE

File | Load Current Program loads the NC Program in the currently active *Program Edit* window into program execution memory. This command is enabled only when an NC Program file is open in the currently active window and the control is in Auto mode. Each program must be loaded into memory before it can be run. Once a program is loaded, it can be run multiple times without being loaded again. Note that only one program can be loaded at a time.

Some very large programs may require several seconds to load. The following message window will be displayed while the control is busy loading a program:

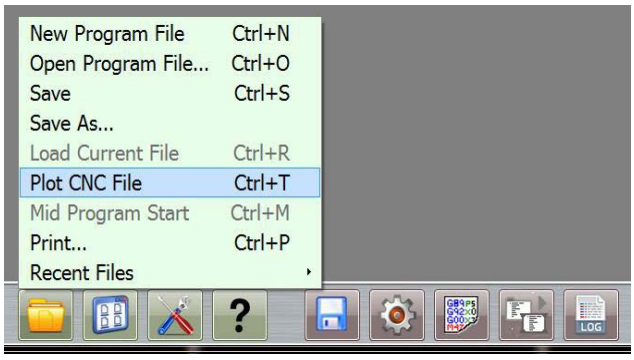
Note: *The System user interface is disabled until the control finishes loading the program.*

Toolbar Shortcut: 

Keyboard Shortcut: ***Ctrl + R***

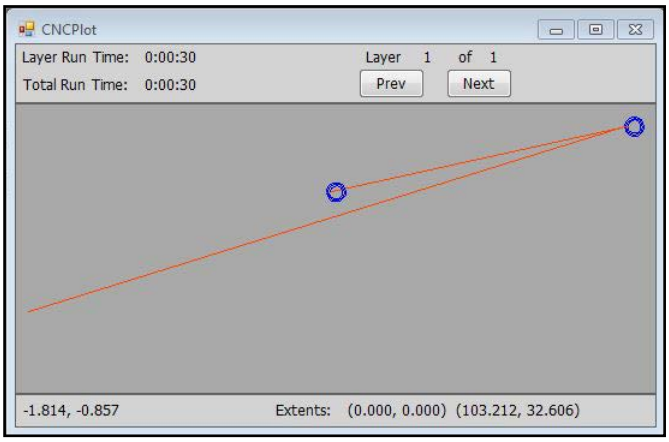
PLOT CNC PROGRAMS

File | Plot CNC File command is an *NC Program Edit* window, and can be used to display a graphical plot of the program, one layer sheet at a time.



If the program has no syntax errors, this function will open a new *Program Plot* window showing the programmed tool path. The plot represents where the head would move if the operator ran the program. If the program has a syntax error, selecting "Plot NC Program" will only display the error message, and the cursor in the *Program Edit* window will be positioned on the line containing the error.

When the program has more than one layer, the “Next” and “Prev” buttons change the plot window to display another layer.



The top section of the *Program Plot* window displays the estimated run time for the current layer and for the entire program. The estimated run time does not account for the effect of the Feedrate Override setting.

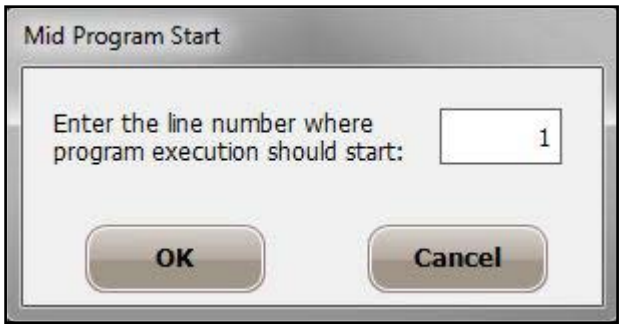
At the bottom of the *Program Plot* window, a status bar displays the overall X-axis and Y-axis dimensions of the program.

To change the plot window magnification, use the **Zoom** functions (see **WINDOW ZOOMING** in the **VIEW MENU** section).

Keyboard Shortcut: ***Ctrl + T***

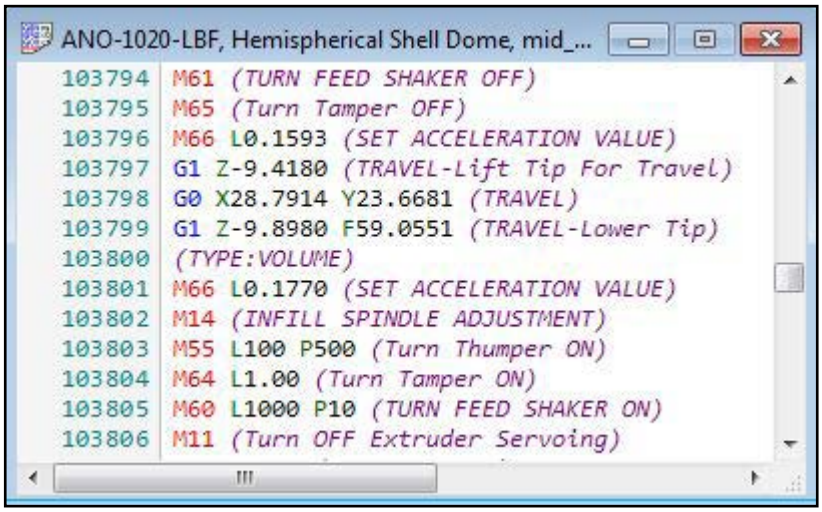
MID PROGRAM START

File | Mid Program Start activates the **Program Restart** function, which allows a program to be restarted at a specified line number. This function is intended for use when programs are terminated before completion and the normal program recovery method cannot be used. When this command is selected, the ***Mid Program Start*** window will be displayed, and prompts the user for an absolute line number to restart the program at.

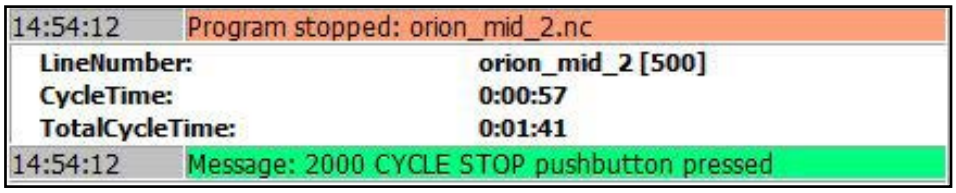


Note: *Absolute line numbers identify the sequential position of a row or block of code in a program file. This number is always relative to the first line in the program, and is not necessarily the same as the optional program line number, “Nxxxxx”.*

To determine the correct line number to restart, line numbers can be turned on in the CNC file editor window. To do this, right click in the window and select the “Show Line Numbers” option. The line numbers will be shown on the left side of the window:



Another way to determine the correct line number to restart is to look at the Log file entry when the program was stopped. Any time a program is stopped, a entry is made in the log file showing the time of the stoppage and the line number that was executing. In the example below, the program stopped while it was in the main program executing line 500:



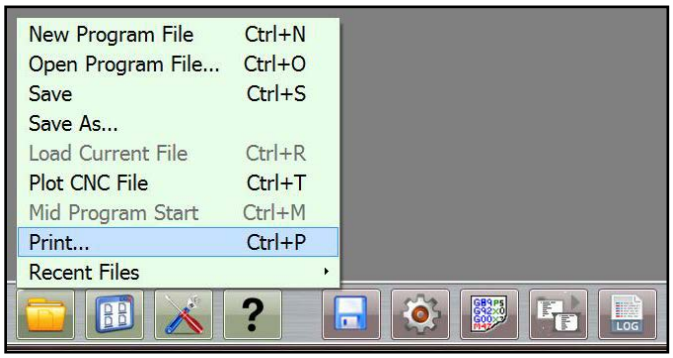
Selecting "Cancel" will close the dialog and abort the restart operation. Selecting "OK" will cause the control to search through the program from the beginning until it finds the desired line number. A message window will be displayed, stating: "Please wait while restart line number is found". This may take a few seconds to finish. If the line number is not reached before an **M30**, **M02**, or **M99** in a main program, an error will be displayed stating "Mid Program start line number not reached".

After the line number is reached, the *CNC Run Window* will be updated to show the next line to be executed. Press "CYCLE START" and the Z-axis will move to the full up position, then the X- and Y-axes will move from the current position to the beginning of the current block at 500 IPM. Normal program execution will resume from there. The following restrictions apply to the **Program Restart** function:

- The **Program Restart** function works best with "straight line" programs, for example, programs that do not contain subroutine calls, macro calls, or "while" loops.
- A program cannot be restarted inside a macro.
- A program can be restarted inside a subroutine as long as that subroutine is located in the same file as the main program. Note, however, if this is done, the program will restart at the first instance the subroutine is called.
- If a program is restarted inside a "while" loop, the program will restart at the first iteration through the loop.
- If the work coordinate system of a program depends on the starting position of the head (for example, a program beginning with **G92 X0 Y0**), then the program will not restart at the correct work coordinates unless the head has not moved since the program was terminated.

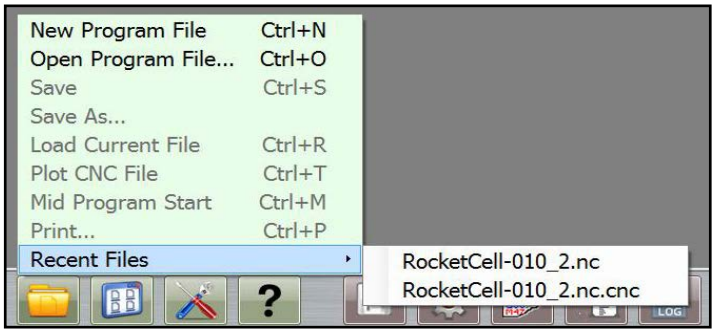
Print

File | Print uses the specified printer to print the document displayed in the active window.



MRU FILE LIST

When closing an NC Program file, that file name is added to the Most Recently Used (MRU) File List. This list, which is displayed near the bottom of the *File* menu, can contain up to ten file names. To quickly reopen a file that was recently closed, simply select the corresponding item from the MRU File List.

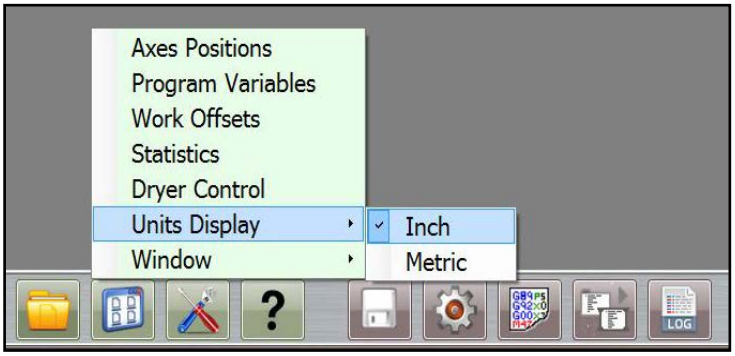


VIEW MENU

The *View* menu allows the user to change how the user interface is displayed and configured, and open various status display windows.

UNITS DISPLAY

View | Units opens a fly-out menu to configure the user interface for displaying English or Metric units.



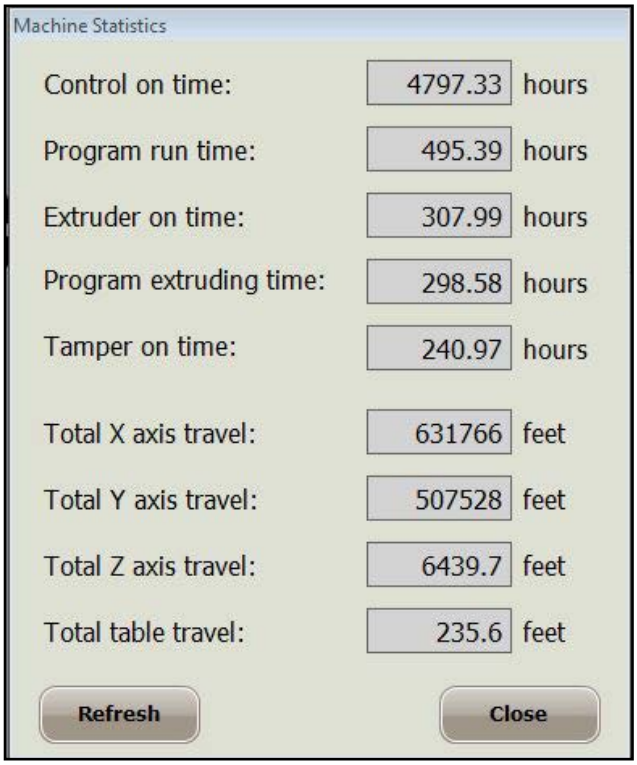
When this command is selected, the *English/Metric* menu appears with a check mark beside the currently selected display units. This selection does NOT affect program execution in inch mode (**G20**) or metric mode (**G21**).

AXES POSITIONS

View | Axes Positions opens the *Axes Positions* window, which is used to monitor the current position of the head in Machine or Work coordinates. The displayed positions change when any axis moves.

STATISTICS

The **View| Statistics** command opens the *Machine Statistics* window. Use this window to view statistical information pertaining to the BAAM System. All data values are initially read-only. However, most values can be changed (for resetting purposes) by Administrator and Manager users.



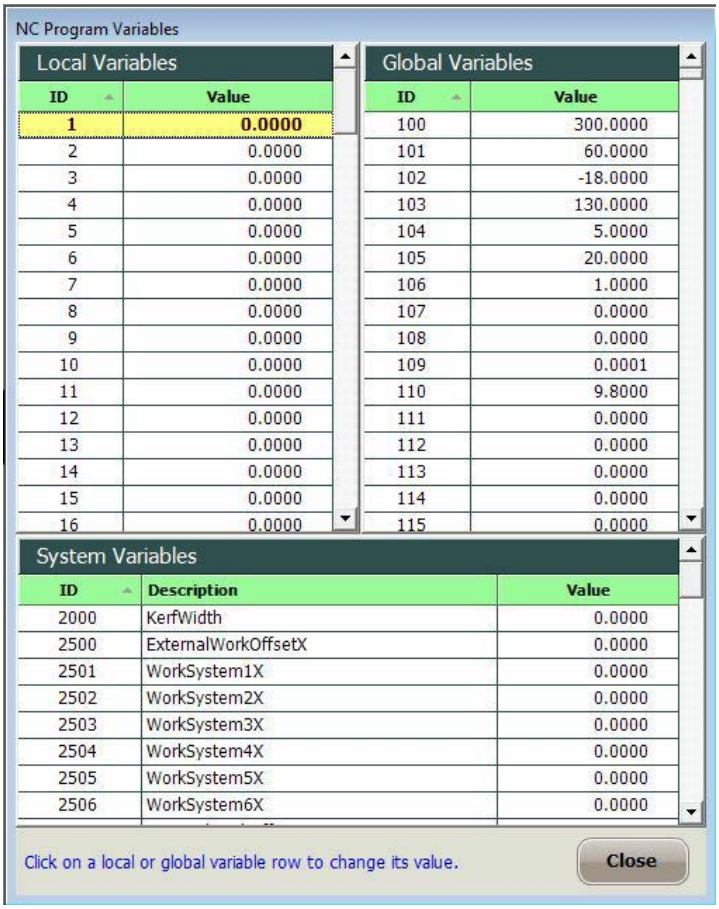
Control on time: Total accumulated time that the control has been powered on and the HMI software has been running.

Program run time: Program execution time. This counter begins when "CYCLE START" is pressed, and stops when program execution is terminated by "CYCLE STOP", **M30**, system alarm, etc.

Extruder on time:	The cumulative time the extruder has been turned on. This includes time in Manual mode and in Auto mode.
Program extruding time:	The cumulative time the extruder has been on while a program is running. This does not include time that that extruder is on in Manual mode.
Tamper on time:	The total amount of time that the Tamper has been on in both Manual mode and Auto mode.
Total X Axis travel:	The cumulative linear distance traveled by the X axis
Total Y axis ravel:	The cumulative linear distance traveled by the Y axis.
Total Z axis travel:	The cumulative linear distance traveled by the Z axis.
Total table travel:	The cumulative linear distance traveled by the table.

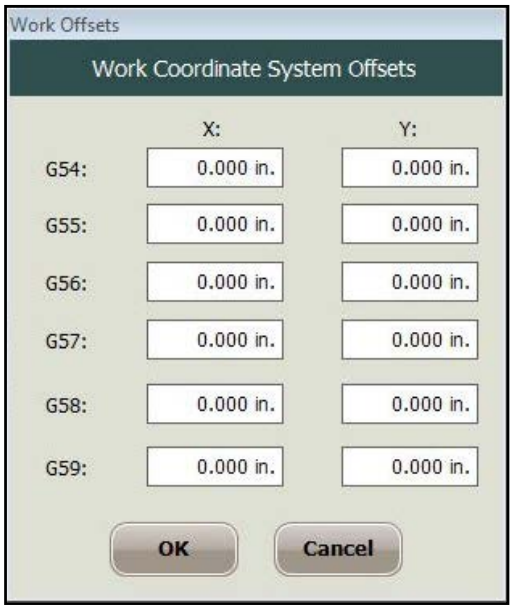
VARIABLES MENU

The *Variables* menu commands provide access to windows that display variables and offset information used by NC programs.



WORK OFFSETS

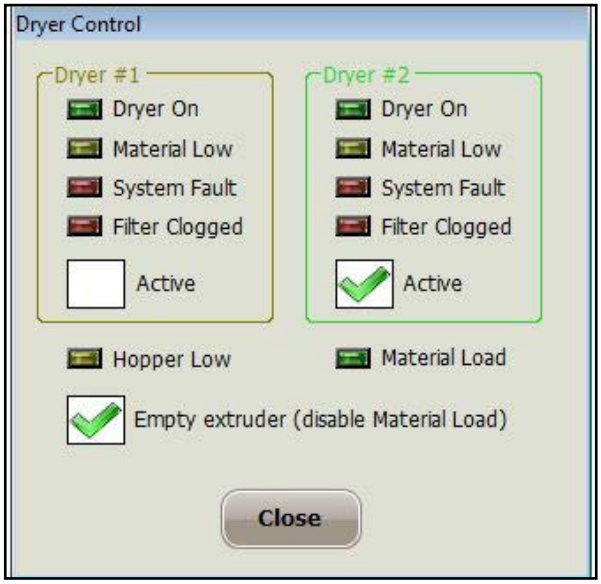
View | Work Offsets allows the user to view or edit the offsets for work coordinate systems 1 through 6 (**G54** to **G59**) and the External Work Offsets. The user can edit work coordinate offsets only when a program is not executing. (See **G54… G59 Work Coordinate System Selection** in the **NC PROGRAMMING** topic.)



Keyboard Shortcut:{ Alt, A, W} sequence

DRYER CONTROL

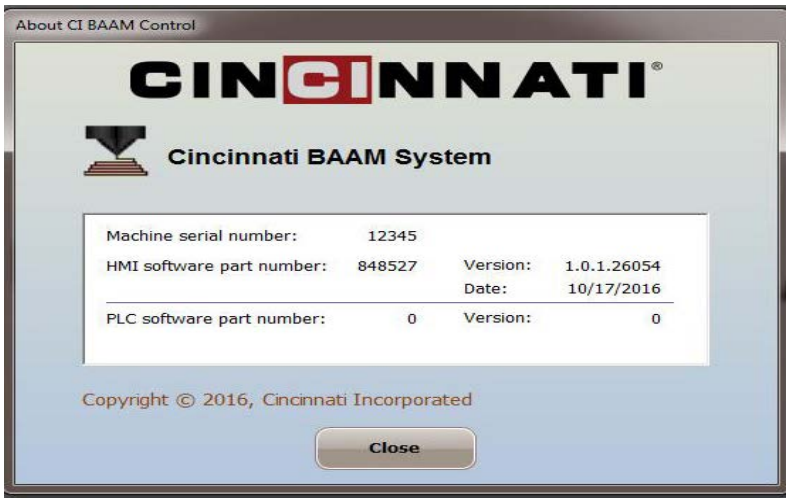
The **View | Dryer Control** command opens the Dryer Control window. This window is used to check the status of the material dryer(s) and the material feed system.



For BAAM systems without the dual dryer option, only Dryer #1 status will be shown. The Active checkbox is only visible on dual dryer systems and is used to select which dryer to use. The Empty Extruder checkbox can be used to temporarily disable the feeding of material from the dryer to the extruder. This can be useful when changing materials to clear the current material from the system before loading the new material.

ABOUT CINCINNATI BAAM SYSTEM

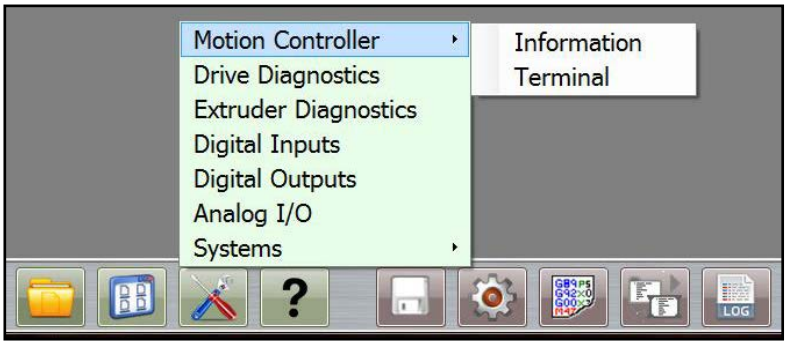
The **Help | About CINCINNATI BAAM System** command opens the *About CNC Control* window, which displays the System's machine model and serial number, and control software version information.



MAINTENANCE MENU

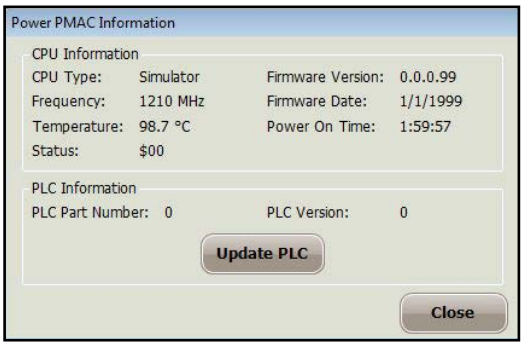
Use the *Maintenance* menu for access to various machine maintenance and diagnostic functions.

MOTION CONTROLLER



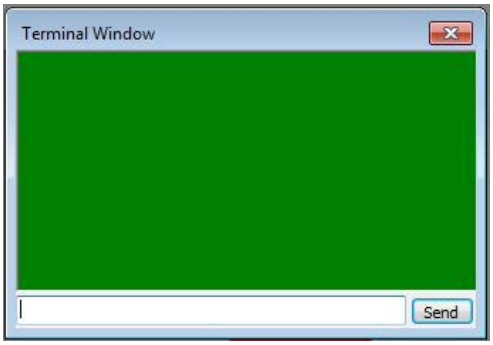
Information

This window shows information about the Power PMAC and the PLC loaded in the Power PMAC.



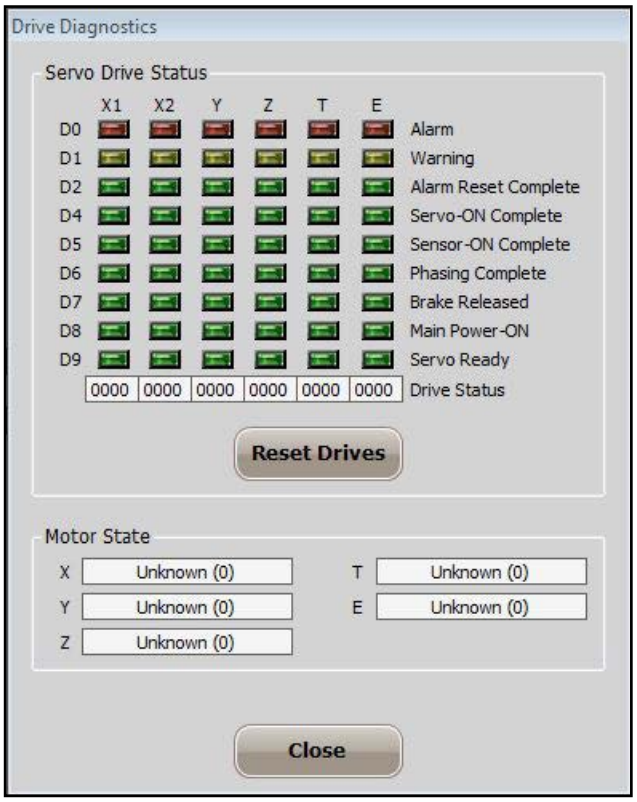
Terminal

This window provides a port into the Power PMAC to allow commands to be sent to the Power PMAC.



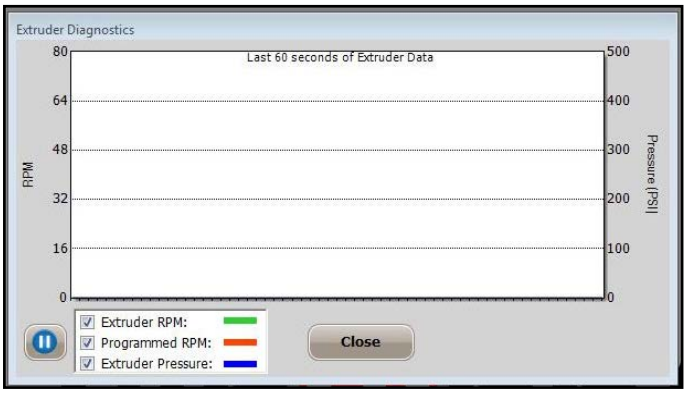
DRIVE DIAGNOSTICS

This window shows information about the servo drives Status, Warnings, and Faults. The servo drives can be reset from this window to clear persistent faults that require a power cycle.



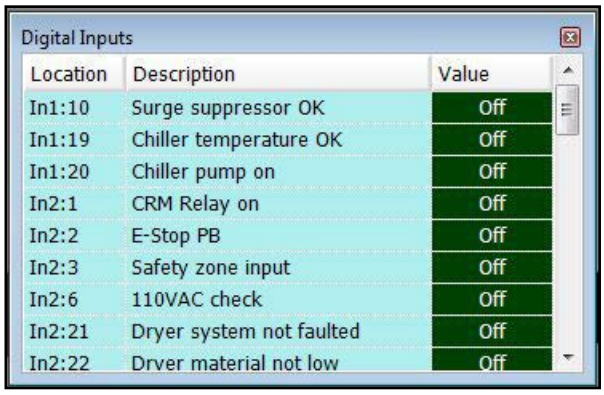
EXTRUDER DIAGNOSTICS

This window shows information about the Extruder process. The last 60 seconds of extruder command RPMs, programmed RPMs and nozzle pressure are shown. The data sets can be selected or de-selected to determine which are displayed in the graph. The graph will auto scale the view. The graph can be paused to allow review of the process.



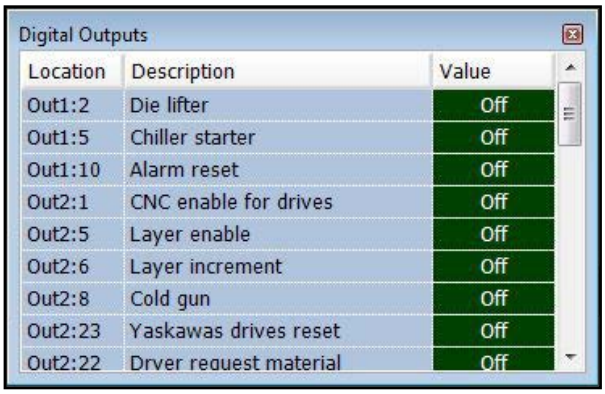
DIGITAL INPUTS

This window shows the current value of all the digital inputs on the BAAM system.



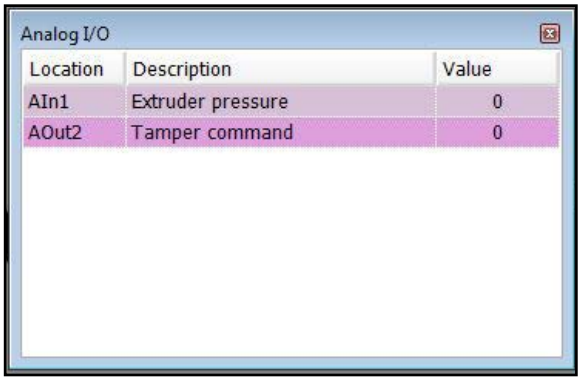
DIGITAL OUTPUTS

This window shows the current value of all the digital outputs on the BAAM system.



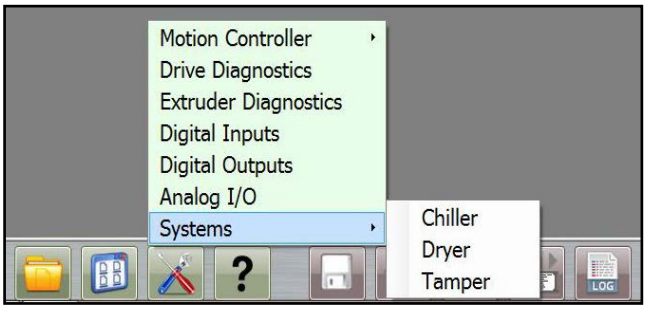
ANALOG I/O

This window shows the current value of all the analog inputs and outputs on the BAAM system.



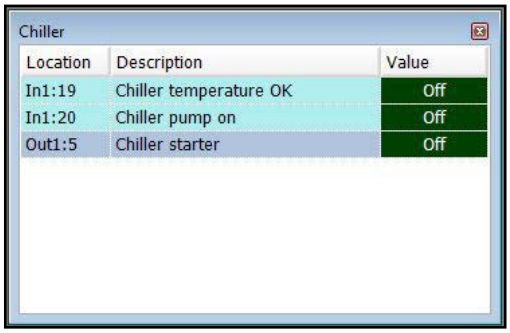
SYSTEMS

The Systems display also be accessed through the Maintenance icon and contains the Chiller, Dryer and Tamper displays. These can be useful to troubleshoot a particular subsystem of the machine.



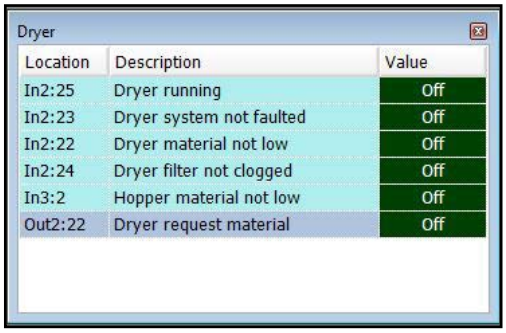
Chiller

The Chiller window



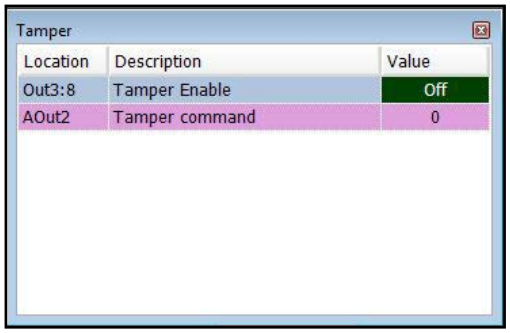
Dryer

The Dryer window



Tamper

The Tamper window



RUNNING A PROGRAM

Starting a Program

1. (Main Drives ON) When the CINCINNATI BAAM HMI application begins running , the *Alarms and Messages* window will display: “Main Drives are off”, “Axes have not been homed” and any other fault messages. Make sure all personnel and equipment remain clear of the moving gantry.

   Turn main drives on by:

   - Press the “RESET” button to clear any other fault messages. Any fault messages that did not clear will need to be corrected. See ALARMS and MESSAGES section.
   - Turn the “DRIVES” selector keyswitch on the Operator Control Station Panel to the “ENABLED” position, press the “DRIVES ON” pushbutton.
   - The “Drives On” indicator on the Control Station Panel should illuminate and the *Main drives are off message* should disappear from the *Alarms and Messages* window.

2. (Homing) The Alarms and Messages window will display: “Axes have not been homed”. Make sure all personnel and equipment remain clear of the moving gantry.

   Home the machine axes by:

   - Selecting the “Home Axes” button.
   - Then pressing the “CYCLE START” pushbutton. When the homing operation starts, the “Home Axes” button text changes to “Homing “ and the color changes to amber.
   - When all axes have moved to their reference positions, all axis motion stops, the Axes home function ends, and the “Home Axes” button will be illuminated green.

3. (Build Surface) The build surface should be clean and debris free. Use build sheets as needed, based on material to be printed. Build sheets should be attached to the build plates to keep the sheet from warping.

4. (Purge Material) When the extruder is at its desired temperatures and the material is loaded in the receiver hopper, then the material can be purged through the extruder.

   - Select manual mode for the machine mode.
   - Jog the axes to put the extruder over the purge collection container.
   - Press “Setup/Test” button. The Setup/Test window will pop-up. In the window there will be extruder controls.
   - Set the extruder speed to zero RPMs.
   - Press the “Extruder” button to turn on the extruder. The “Extruder” button will light green when the extruder is running.
   - Slowly increase the extruder RPMs, allowing the material to begin to flow down the extruder.
   - Once the material begins to steadily flow out the nozzle, then raise the RPMs to the desired print speed. Verify the quality of the bead and adjust the RPMs or extruder temperatures as needed.
   - Press the “Extruder” button to turn off the extruder. The “Extruder” button’s light green will be off when the extruder is not running.

5. (Load Program) Once the machine has been homed, the table temperature is reached and the material has been loaded in the extruder, the machine is ready to print. Follow the above procedures to Open and Load a program. The program

will be checked, correct any program errors as needed.

6. (Run Program) When the machine is ready to print and the file to be printed is loaded, start the print.
   - Select Auto mode for the machine mode.
   - Verify the program to be printed.
   - Press the “Cycle Start” button to start the machine. You may be asked to verify the operation and be required to hit the “Cycle Start” button multiple times to get the printing started.

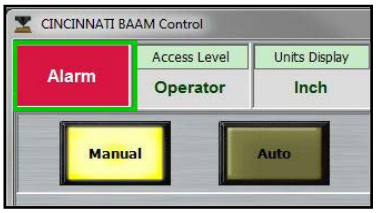
RESTARTING A PROGRAM - ERROR RECOVERY

Follow these steps to resume an interrupted program at the point where the interruption occurred.

1. Correct fault or stopping condition.
2. Put the control in Auto mode by pressing the “Auto” button on the Operator Console Control Bar.
3. Activate Program Test mode by pressing the “Program Test” button on the Program Modes Control Bar.
4. Press the “CYCLE START” button on the Control Station front panel. The extruder will return to the position at the beginning of the program block that was being executed when the interruption occurred. If the interrupted block did not contain any motion commands, the head will not move when “CYCLE START” is pressed.
5. Let the nozzle move through the program within a few inches of the point of interruption. At this point, deselect the Program Test option. This will prompt the extruder to begin expelling material again.

ALARMS AND MESSAGES

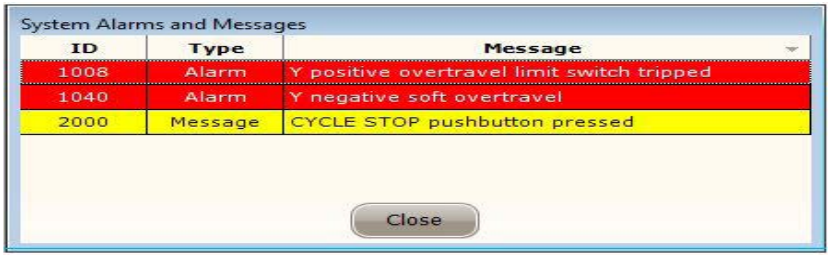
The System Alarms and Messages window will be displayed automatically when a new Alarm occurs or it can be manually opened by clicking in the Alarm status window in the upper left corner.



The Alarm status window will look like this when a System Alarm is active:

And it will look like this when only a Message is active:

It will be blank if no alarms or messages are active



OPERATOR FYI MESSAGES

MESSAGE NUMBER	DISPLAYED MESSAGE	DESCRIPTION	POSSIBLE CAUSE(S)
2000	CYCLE STOP pushbutton pressed.	One or more of the CYCLE STOP or FEEDHOLD buttons are pressed.	If buttons are not being pressed: • Bad wire connection to button.
2002	Remote station is enabled - no axis or table moves permitted from console.	Remote pendant key is in the ENABLE position.	Disable Remote Station.
2003	Feedrate override is set to 0%.	Feedrate is manually set to 0% on UI.	Correct Feedrate Override.
2011	Y encoder loss of counts detected.	The encoder that tracks the Y plate motion has lost counts.	Inspect the encoder tape, home axis, review maintenance section on encoders.
2012	Z encoder loss of counts detected.	The encoder that tracks the Z motion has lost counts.	Inspect encoder set screws, home axis, review maintenance section on encoders.
2015	Motors are phasing.	Phasing of linear motors.	Homing machine.
2016	Table encoder loss of counts detected.	Table encoder fault.	Check table encoder.
2017	Invalid machine mode.		
2018			
2019	Extruder pressure warning!	Screw pressure is above set limits.	Adjust system settings to bring pressure within allowable range.
2020	Drive Reset in Progress.	Resetting drive to clear fault or changing setup.	
2021	Dryer system faulted.	System faulted.	See dryer documentation.
2022	Dryer low material level.	Material supply is at low level.	Add material.
2023	Dryer filter clogged.	The air resistance is too great as it passes through the filter.	Inspect / Clean filter. Confirm proper operation of the filter monitoring system.
2024	Dryer not running.	Power is OFF to the dryer.	Determine why power is OFF to the dryer.
2025	Hopper low material/empty.	Material is not feeding or not feeding fast enough.	Adjust feed time or position of proximity switch at the receiver location. Check for material restriction in the line or dust bags.
2026	Material loading is disabled. See Dryer Control window to enable.	Material Loading has been disabled through the BAAM UI.	Enable Material Loading in the UI. View/ Dryer Control.
2027	X Axis is disabled!	Axis has been disabled in machine Configurations.	Enable Axis in Machine Configurations.
2028	Y Axis is disabled!	Axis has been disabled in machine Configurations.	Enable Axis in Machine Configurations.
2030	Table is disabled!	Axis has been disabled in machine Configurations.	Enable Axis in Machine Configurations.
2031	Extruder is disabled!	Axis has been disabled in machine Configurations.	Enable Axis in Machine Configurations.

SYSTEM ALARMS

MESSAGE NUMBER	DISPLAYED MESSAGE	DESCRIPTION	POSSIBLE CAUSE(S)
1000	PLC timer overrun.	[Failure] Internal software error. Contact CINCINNATI.	
1001	CNC watchdog failure.	[Failure] PC and motion controller experienced a break in communications.	
1002	Console input board failure	[Failure] The console I/O board is no longer communicating with the CNC control. (If this message is displayed, ignore all other System Alarms until this message is cleared.)	• 24VDC power supply failure. • Faulty connection in flat 10-conductor I/O board cable (console or PC enclosure). • Faulty connection at I/O communications board.
1003	Console output board failure	[Failure] See “Console input board failure “.	• See “Console input board failure
1004	X1 Positive overtravel limit switch tripped	[Normal] The X1 positive overtravel limit switch was actuated.	• Faulty limit switch • Faulty wiring • Check connection at Drive Amp interface board
1005	X1 negative overtravel limit switch tripped	[Normal] The X1 negative overtravel limit switch was actuated.	• Faulty limit switch • Faulty wiring • Check connection at Drive Amp interface board
1006	X2 positive overtravel limit switch tripped	[Normal] The X2 positive overtravel limit switch was actuated.	• Faulty limit switch • Faulty wiring • Check connection at Drive Amp interface board
1007	X2 negative overtravel limit switch tripped	[Normal] The X2 negative overtravel limit switch was actuated.	• Faulty limit switch • Faulty wiring • Check connection at Drive Amp interface board
1008	Y positive overtravel limit switch tripped	[Normal] The Y positive overtravel limit switch was actuated.	• Faulty limit switch • Faulty wiring • Check connection at gantry interface board • Check connection at Drive Amp interface board
1009	Y negative overtravel limit switch tripped	[Normal] The Y negative overtravel limit switch was actuated.	• Faulty limit switch • Faulty wiring • Check connection at gantry interface board • Check connection at Drive Amp interface board
1011	Surge suppressor tripped	The surge suppressor has detected problems with the incoming power (I.e. loss of phase, over or undervoltage) or has detected a loss of protection in it’s circuits.	• Check incoming power and power and fuses. Replace suppressor if necessary.
1012	Motors have not been phased	Homing not complete.	Homing machine
1013	Delta Tau Macro Ring error		
1014	Emergency stop pressed	[Failure] E-Stop pressed.	• Faulty pneumatic or electrical circuit connection. • Gate is jammed or obstructed.
1016	Z-axis motion timed out		
1021	Main drives are off	[Normal]	Turn on Main Drive Switch and press drives on button
1022	Chiller fault		

MESSAGE NUMBER	DISPLAYED MESSAGE	DESCRIPTION	POSSIBLE CAUSE(S)
1023	Process interrupted by CYCLE STOP or RESET button	[Normal] A CYCLE STOP, FEEDHOLD, or RESET button was pressed during one of the following: • Homing. • Part program execution.	If no button was pressed: • Bad wiring connection to a CYCLE STOP or FEEDHOLD button located on the console or hand-held pendant.
1024	Drive phasing error - press “Reset Drives” button		
1025	Drive error - check amplifier status		
1026	X motor parameters don’t match	[Failure] X1 and X2 speed and acceleration parameters must be identical. (Necessary to maintain gantry alignment when moving.)	Error in parameter file.
1027	Motor X1 thermostat	[Failure] Motor thermostat contact has opened, indicating that the motor has overheated.	• Obstruction in coolant lines to motor. • • If thermostat contact is not open: • • Bad wire connection from motor thermostat to • Board. • • I/O Board failure has occurred (see alarm • description). • • Power failure.
1028	Motor X2 thermostat	[Failure] See “Motor X1 thermostat”.	• See “Motor X1 thermostat”.
1029	Y motor thermostat	[Failure] Motor thermostat contact has opened, indicating that the motor has overheated.	• Obstruction in coolant lines to motor. • • If thermostat contact is not open: • • Bad wire connection from motor thermostat to • I/O PC Board. • • I/O Board failure (See alarm description). • • Power failure.
1034	Retract head before moving axis		
1035	X1 positive soft overtravel	[Normal] The position of the motor exceeded the Overtravel Limit setting on the Configuration screen or a hard overtravel limit switch was actuated.	• If occurring during JOG mode: • • Overtravel limits not correctly set on • Configuration screen. • • Clamp Zone parameters not set correctly on • Configuration screen. • If occurring during AUTO mode: • • Part program commanding X/Y too close to • end of travel
1036	X1 negative soft overtravel	[Normal] The position of the motor exceeded the Overtravel Limit setting on the Configuration screen or a hard overtravel limit switch was actuated.	• If occurring during JOG mode: • • Overtravel limits not correctly set on • Configuration screen. • • Clamp Zone parameters not set correctly on • Configuration screen. • If occurring during AUTO mode: • • Part program commanding X/Y too close to • end of travel.
1037	X2 positive soft overtravel	[Normal] See X1 positive soft overtravel	• See X1 positive soft overtravel.
1038	X2 negative soft overtravel	[Normal] See X1 negative soft overtravel.	• See X1 negative soft overtravel.
1039	Y positive soft overtravel	[Normal] See X1 positive soft overtravel.	• See X1 positive soft overtravel.
1040	Y negative soft overtravel	[Normal] See X1 negative soft overtravel.	• See X1 negative soft overtravel
1041	X1 amplifier fault	[Failure] The X1 amplifier has sensed a malfunction and has disabled itself. (Drives and High Voltage are disabled.)	• Motor wiring has been shorted. • • Amplifier has overheated due to continuous • movement in RAPID mode. • • Bad wiring connection between amp and • UMAC.
1042	X2 amplifier fault	[Failure] See X1 amplifier fault.	• See X1 amplifier fault.
1043	Y amplifier fault	[Failure] See X1 amplifier fault.	• See X1 amplifier fault.

MESSAGE NUMBER	DISPLAYED MESSAGE	DESCRIPTION	POSSIBLE CAUSE(S)
1044	X1 fatal following error	[Failure] The motor did not go where it was commanded. The actual position of the motor may not differ from the commanded position. Drives and High Voltage are disabled.	• Incorrect Following Error Limit parameter on the • Configuration screen. • • X or Y “Jog/Rapid Acc/Dec Rate” set too high on • Configuration Screen. • • 3-phase input voltage to machine too low. • • Faulty encoder. • • Faulty wiring on motor or encoder. • • Improperly installed magnet track (magnets out • of order or improper gap). • • Debris on magnet track. • • An excessive feedrate was commanded for a • long contour move.
1045	X2 fatal following error	[Failure] See X1 fatal following error.	• See X1 fatal following error.
1046	Y fatal following error	[Failure] See X1 fatal following error.	• See X1 fatal following error.
1047	Safety gate open	[Normal] The cut area enclosure gate is open.	• If gate is not visibly open: • • Must press RESET to reset safety relay. • • Gate switch is not made. • • Both doors must be closed to clear alarm.
1048	Axis amplifiers are not enabled - press RESET to re-enable	[Normal] The RESET button must be pressed to reactivate the X1, X2 and Y-amplifiers, following a loss of power to the DRIVES. (DRIVES power must be reestablished before this message will appear.)	• The DRIVES selector switch was turned to • “LOCK”. • • An Auto Shutdown occurred, which • automatically causes DRIVES power to turn • off. • • An X, Y or chiller fault occurred, which • automatically causes DRIVES power to turn • off.
1049	Axes have not been homed	[Normal] X and Y-axis motors must be “phased” to the magnet track and homed before continuing.	
1050	Remote station must be enabled		
1051	Remote station must be disabled		
1052	Internal PMAC error: Run time	[Failure] Motion controller run-time error.	
1053	Internal PMAC error: Servo	[Failure] Motion controller servo error.	
1054	Internal PMAC error: Circle radius	[Failure] Part programming error. Ending point specified for arc/circle cannot be reached using specified radius.	
1055	I/O Board #1 failure		
1056	I/O Board #2 failure		
1057	I/O Board #3 failure		
1058	I/O Board #4 failure		
1059	110 volt AC power failure	[Failure] The 110 VAC monitoring relay has detected a loss of power.	
1061	Chiller temperature out of range	[Failure] Chiller unable to maintain temperature within 5 degrees of set point.	• Filters dirty/clogged. • • Side panel not installed on chiller. • • 1 or more fans not running
1064	Z amplifier fault	[Failure] See X1 amplifier fault.	• See X1 amplifier fault.
1065	Z axis not homed	[Normal] The Z-axis has not successfully performed a homing move since the last time the control application was started.	• If the Z-axis will not home: • • Z-home proximity switch malfunction or bad • connection. • • Z-down proximity switch malfunction or bad • connection.
1066	Z axis fatal following error	[Failure] The motor did not go where it was commanded to go. The actual position of the motor may not differ from the commanded position.	• Incorrect Following Error Limit parameter on • the Configuration screen. • • Faulty encoder or brake. • • Faulty wiring on motor or encoder.

MESSAGE NUMBER	DISPLAYED MESSAGE	DESCRIPTION	POSSIBLE CAUSE(S)
1067	Z axis positive soft overtravel	[Normal] The Z-axis Machine Position has exceeded the Overtravel limit setting on the Configuration screen.	• Positive Overtravel Limit parameter not set • correctly on Configuration screen.
1068	Z axis negative soft overtravel	[Normal] The Z-axis Machine Position exceeded the Negative Overtravel limit setting on the Configuration screen	• Incorrect Negative Overtravel Limit parameter on Configuration screen.
1069	Z axis amp not enabled - press RESET	[Normal] The RESET button must be pressed to reactivate the Z-amplifier. This message will always appear following loss of DRIVES power, or certain Z-axis faults. (DRIVES power must be re-established before this message will appear.)	• The DRIVES selector switch was turned to • “LOCK/OFF”. • • An Auto Shutdown occurred, which • automatically turns off DRIVES power. • • An X, Y or chiller fault occurred, which • automatically turns off DRIVES power. • • One of the following alarms occurred: Z-fatal • following error, Z-amp fault, Z-soft overtravel. • If Z-axis will not home: • • Both the Z-home and Z-down proximity • switches are tripped, indicating bad switch • operation or wiring.
1070	Z axis non-contact hard overtravel	[Normal] The Y-plate separated from the Z-block during noncontact tracking.	• Z-Down proximity switch out of adjustment. • • Bad wire connection on Z-Down proximity • switch.
1071	Table amplifier fault	[Failure] See X1 amplifier fault.	See X1 amplifier fault.
1072	Table not homed	[Normal] The Table axis has not successfully performed a homing move since the last time the control application was started.	If the Table axis will not home, Check ABS encoder malfunction or bad connection.
1073	Table fatal following error	[Failure] The motor did not go where it was commanded to go. The actual position of the motor may not differ from the commanded position	Incorrect Following Error Limit parameter on • The Configuration screen. • Faulty encoder or brake. • Faulty wiring on motor or encoder. • Z-axis “Max Speed for all Moves” too high
1074	Table positive soft overtravel	[Normal] The position of the motor exceeded the Overtravel Limit setting on the Configuration screen or a hard overtravel limit switch was actuated.	See X1 positive soft overtravel.
1075	Table negative soft overtravel	[Normal] The position of the motor exceeded the Overtravel Limit setting on the Configuration screen or a hard overtravel limit switch was actuated.	See X1 negative soft overtravel.
1076	Table amp not enabled - press RESET	[Normal] The RESET button must be pressed to reactivate the Z-amplifier. This message will always appear following loss of DRIVES power, or certain Z-axis faults. (DRIVES power must be re-established before this message will appear.)	
1077	Table timed out before reaching position		
1078	Motion controller run time error:		
1079	Extruder pressure too high	Extruder Barrel pressure to high.	Reduce extruder RPMs or adjust backpressure screw.
1080	Extruder amplifier fault		
1081	Extruder not homed	[Normal] The Extruder axis has not successfully performed a homing move since the last time the control application was started.	If the Extruder axis will not home: • Check encoder malfunction or bad connection
1082	Extruder fatal following error	Failure] The motor did not go where it was commanded to go. The actual position of the motor may not differ from the commanded position.	

MESSAGE NUMBER	DISPLAYED MESSAGE	DESCRIPTION	POSSIBLE CAUSE(S)
1083	Extruder positive soft overtravel	[Normal] The position of the motor exceeded the Overtravel Limit setting on the Configuration screen or a hard overtravel limit switch was actuated.	There is no switch check wiring for jumper.
1084	Extruder negative soft overtravel	[Normal] The position of the motor exceeded the Overtravel Limit setting on the Configuration screen or a hard overtravel limit switch was actuated.	There is no switch check wiring for jumper
1085	Extruder amp not enabled - press RESET		
1086	Extruder timed out before reaching position		

TROUBLESHOOTING

Effective and safe troubleshooting procedures are acquired through experience and a thorough knowledge of the machine and its operation. The use of maintenance instructions, assembly drawings, and schematics included with the System **Operation, Safety, and Maintenance Manual** will be helpful in resolving problems with the machine.

The following chart was developed to aid in troubleshooting problems with the machine. The chart contains questions frequently asked by our customers. If major repairs are required or if the problem has not been identified by using the chart, contact the Service department at CINCINNATI INCORPORATED for assistance. Only qualified personnel should attempt to troubleshoot electrical systems.

PROBLEM	POSSIBLE CAUSE	REPAIR
Control will not turn on.	No power to machine.	• Verify line voltage. • Check condition of fuses / disconnect.
	Power supply failure.	• Verify power supply voltage. • Check condition of fuses.
	UPS (uninterruptible power supply) turned off.	• If the UPS was turned off manually, it must be turned on before the control will start.
Cannot start cycle.	Alarm condition present. Improper control configuration.	• Check alarm status and correct any conditions present. • Verify correct operating mode and program status.

NC PROGRAMMING

STANDARD G CODES

G codes are used for program control of specific machine functions...

The following four G codes all move the printing nozzle to commanded Work coordinates:

G00 Rapid Traverse move

G01 Linear move

G02 Clockwise Arc

G03 Counterclockwise Arc

These four G codes form a modal group; the last G code commanded in the group is active for all blocks until the program commands another G code in the group. The default code when a program starts is **G00**. The leading zero can be omitted; **G0**, **G1**, **G2**, and **G3** are the same as **G00**, **G01**, **G02**, and **G03**.

Each of these G codes specifies the end of the move with X and Y in the Work coordinate system. X and Y are absolute coordinates when the program commands the block in **G90** mode, and incremental distances when commanded in **G91** mode. The command must specify at least one axis.

The **G00** command moves the axes at the rapid traverse rate of the machine. **G01**, **G02,** and **G03** move the axes at the contouring feedrate (optionally specified in the block with **F**). When the block does not command a feedrate, the program uses the last defined contouring feedrate. When the control applies the rapid traverse rate for a **G00** move, it does not change the contouring feedrate used by the **G01**, **G02**, and **G03** blocks.

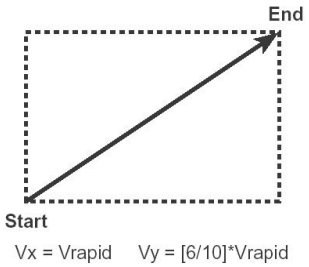
G00 RAPID TRAVERSE MOVE

G00 X__ Y__ Rapid Traverse move

The **G00** command moves the printing nozzle to a work coordinate location (or incremental distance) using the rapid traverse rate. When the command requires both axes to move, the axis moving the longer distance uses the rapid traverse rate of the machine. The other axis moves at a lower velocity proportional to the distance required, so both reach their endpoints at the same time, approximating linear interpolation.

Example:

(G91) G00 X10 Y6

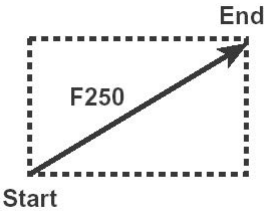


G01 LINEAR MOVE

G01 X__ Y__ (F__) Linear move

This command moves the head to the work coordinates (or incremental distance) defined by X and Y, at a contouring feedrate optionally specified by **F**.

Example: **(G91) G01 X6 Y4 F250**



When the command requires both axes to change position, the machine moves each axis at a velocity required to produce a combined feedrate equal to the contouring feedrate. The move follows the linear path between start and end points.

G02 AND G03 ARC MOVE

G02 X__ Y__ I__ J__ (F__) Clockwise Arc move (offsets specified)

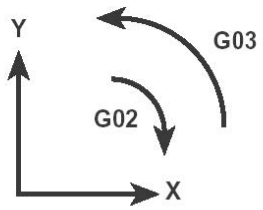
G02 X__ Y__ R__ (F__) Clockwise Arc move, (radius specified)

G03 X__ Y__ I__ J__ (F__) Counterclockwise Arc move, (offsets specified)

G03 X__ Y__ R__ (F__) Counterclockwise Arc move, (radius specified)

A program uses **G02** (clockwise arc) or **G03** (counterclockwise arc) to command a circular contouring move ending at the work coordinates (or incremental distances) specified by X and Y. The command defines the shape of the arc either by specifying incremental distances (with **I** and **J**) from the starting position to the center, or by specifying the radius (with **R**). The control software interprets **I** and **J** as distances in the X and Y directions (respectively) from the starting position to the center. When the command specifies radius **R**, the control moves the nozzle along a circular path with that radius.

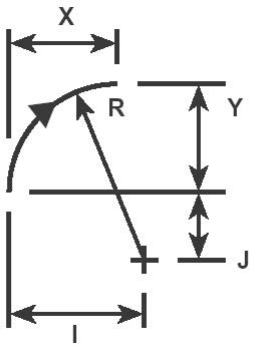
The machine maintains the modal contouring feedrate **(F**) along the circular path.



G02 X_ Y_ I_ J_ (F_)
G02 X_ Y_ R_ (F_)
G03 X_ Y_ I_ J_ (F_)
G03 X_ Y_ R_ (F_)

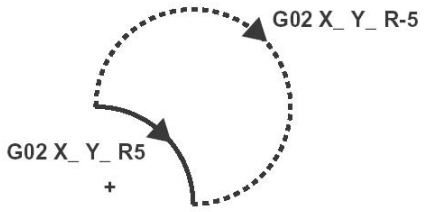
Example:

(G91) G02 X5 Y4 I7 J-3



When the block uses **R** instead of **I** and **J,** there are two possible arcs for a given direction (CW or CCW) and end coordinates. To specify which arc to contour, the block commands **R** with a positive or negative sign. To specify an arc that is less than 180 degrees, the block commands a positive **R** value. To specify an arc greater than 180 degrees, the **G02** or **G03** block commands **R** with a negative value.

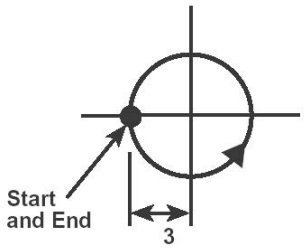
Example:



When **G02** or **G03** specifies the same coordinates for the start and end of the arc, the machine contours a complete circle. For complete circles, the block must specify the center with **I** and **J.** Programming software must specify both coordinates accurately. If the ending coordinates for a circular move are not exactly the same as the starting coordinates, the path may be a very small arc instead of a complete circle. To avoid this problem, programs can omit **X** and **Y** from a **G02** or **G03** block to command a complete circle; the control will automatically apply the same starting and ending coordinates.

Example:

(G91) G03 I3 J0



G04 DWELL

G04 P__ Program Dwell

The **G04** (or **G4**) command causes NC program execution to dwell for the time specified by the **P** argument. If the P argument contains a decimal point, the value is in seconds. If the P argument does not contain a decimal point, the value is in milliseconds.

Example (to dwell for one second): **G04 P1000 or G04 P1.0**

G09 EXACT STOP (ONE BLOCK)

G09 Use Exact Stop for one block

The program commands **G09** (or **G9**) in the same block as a **G00**, **G01**, **G02** or **G03** command. When the block contains **G09**, the control does not proceed to the next block until the axes reach zero feedrate. If the block does not contain **G09**, the control proceeds to the next block when each axis position is within a critical distance of the commanded position.

Example: **(G01 X_ Y_) G09**

G20 INCH MODE, G21 METRIC MODE

G20 Use Inch mode

G21 Use Metric mode

The **G20** command puts the CNC in the inch units mode. In **G20** mode, the control interprets program coordinates and feedrates in inch system units. (Positions are in inches and feedrates are in inches per minute).

The **G21** command puts the CNC in the metric units mode. In **G21** mode, the control interprets program coordinates and feedrates in metric system units. (Positions are in millimeters and feedrates are in millimeters per minute).

The default mode is **G20** when the **CNC/HMI** application starts. After the control runs a program, the default mode is the same as the last program. To make sure the control interprets a program correctly, the program should begin by commanding **G20** or **G21** to specify units.

G20 and **G21** do not change the units mode of the **CNC/HMI** application. All user interface windows display values in inch or metric units as selected by **View | Units**.

G21 Metric mode

Metric mode is specified. All arguments, including X/Y arguments, are expressed in millimeters.

G50 AND G51 COORDINATE SYSTEM SCALING

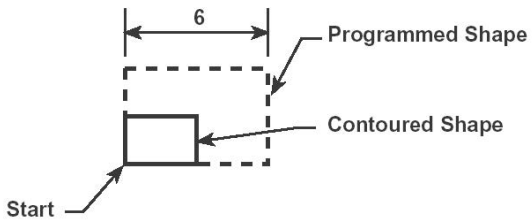
G50	Cancel Work Coordinate System scaling
G51 X__ Y__ P__	Scale Work Coordinate System
G51 X__ Y__ I__ J__	Scale Work Coordinate System

The control interprets the work coordinate system at a different scale or as a mirror image when the program commands **G51**. The program can restore the normal scale by commanding **G50**. When each program starts, the default mode is **G50**. The *Absolute Position* window and system variables indicate the actual position.

The **G51** block defines the center of scaling with X and Y, and the scale factor with **P**, **I** or **J**. To command 1.0 scale (where the contoured shape is the same as the programmed shape), the **G51** block uses **P1000** (or **I1000** or **J1000**). The **G51** block can use **I** and **J** to command separate scale factors for the X- and Y-axes (respectively). To contour a mirror image of the programmed shape, the block commands **I** or **J** with a negative value. The control does not scale the kerf compensation offset distance when the program commands scaling.

Example 1:

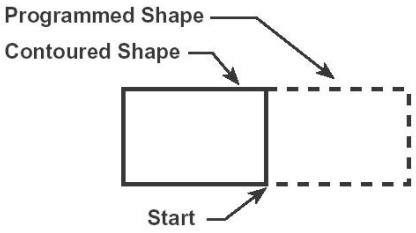
```
G91
G51 X0 Y0 P500
G01 X6
Y4
X-6
Y-4
G50
```



Example 2:

```
G91
G51 X0 Y0 I-1000
G01 X6
```

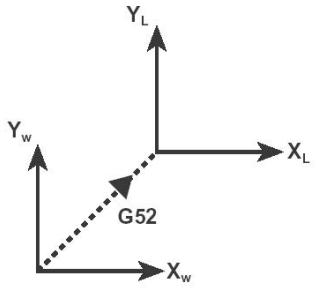
Y4
X-6
Y-4
G50



G52 LOCAL WORK COORDINATE SYSTEM

G52 X__ Y__ Define Local Work Coordinate System

The G52 command temporarily defines a new work coordinate system while remembering the original. The zero position of the new (or "local") coordinate system is at the coordinates in the original coordinate system specified by X and Y in the G52 block. After the G52 block, the program makes contouring moves using the new coordinate system. To restore the original work coordinate system, the program commands G52 X0 Y0.



The G52 block does not move the nozzle. The *Absolute Position* window changes to indicate the nozzle position in the temporary coordinate system.

To demonstrate how a program could use G52, consider a program that uses a subprogram to contour the same shape several times, and both the main program and subprogram use G90 (absolute) mode. The main program would command a work coordinate system with G92 and the subprogram would command a local coordinate system with G52, then cancel it with G52 X0 Y0.

G53 RAPID MOVE TO MACHINE COORDINATES

G53 X__ Y__ Rapid Move to Machine Coordinates (X Y)

The G53 command moves the nozzle at the rapid traverse rate to a position specified by X and Y in the machine coordinate system. G53 is only active in one block and only in G90 absolute mode. No motion occurs if the program commands G53 in G91 (incremental) mode. The control does not change the machine coordinate system when the program commands kerf compensation, rotation, scaling, or mirror image, or if the program changes the work coordinate system.

G54…G59 WORK COORDINATE SYSTEM SELECTION

G54 (Use Work Coordinate System 1)

G55 (Use Work Coordinate System 2)

G56 (Use Work Coordinate System 3)

G57 (Use Work Coordinate System 4)

G58 (Use Work Coordinate System 5)

G59 (Use Work Coordinate System 6)

A program can use **G54** through **G59** to command one of six different pre-defined work coordinate systems. The user can set the distance from Machine X0 Y0 to the Work X0 Y0 position of each coordinate system in the **Work Offsets** window, or the program can assign the distance with system variables #2501 through #2506 (**X**) and #2601 through #2606 (Y).

A work coordinate system defined with **G54** through **G59** does not need **G92** to define its X0 Y0 position. **G54** through **G59** override **G92** by commanding a work coordinate system with its X0 Y0 position preset on the machine.

The **G54** through **G59** block does not move the nozzle. The *Absolute Position* window changes to indicate the nozzle position in the new work coordinate system.

G61 AND G64 EXACT STOP MODE

G61 Use Exact Stop mode

G64 Cancel Exact Stop mode

G61 commands the CNC to use Exact Stop mode. In this mode, the axes decelerate to a stop at the end of every **G00**, **G01**, **G02,** or **G03** block. The CNC remains in **G61** mode until the program commands **G64** or the program ends.

The **G64** command cancels Exact Stop mode. The default mode when each program starts is **G64**. In **G64** mode, the control proceeds to the next block when each axis position is within a critical distance of the commanded position.

G65 SUB-PROGRAM CALL

G65 P__ (A__ B__ C__ D__ etc…) Sub-Program call (with optional arguments)

The **G65** block specifies the subprogram name after **P**, and may use other arguments to set local variables in the subprogram.

The **G65** block must include a **P** argument, followed by the name of the subprogram. If the subprogram is in the same file as the CNC program, then the subprogram name does not need an extension or path. However, if the subprogram is in a separate file then the **G65** block must include a **P** argument followed by the subprogram file name including its extension (if any) and its path if different from the calling program.

If the **G65** command includes optional arguments, the command must have a space between the last character of the program name and the first argument. This is required because program names can contain both numerals and alphabetic characters. For more on calling subprograms with **G65**, see the Program Structure topic.

G68 WORK COORDINATE ROTATION

G68 X__ Y__ R__ Rotate Work Coordinate system

G69 Cancel Work Coordinate System rotation

A program can use the **G68** command to rotate the work coordinate system relative to the machine axes. The command specifies the center of rotation with X and Y work coordinates (or incremental distances). The command specifies the amount of rotation with argument **R** in degrees, with counterclockwise positive. In **G90** mode, **R** is the absolute angle of rotation. In **G91** mode, **R** is the incremental rotation angle that the control adds to any previous rotation.

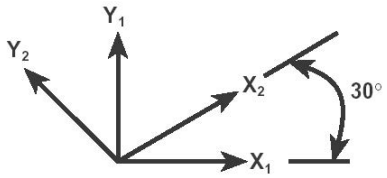
The work coordinate system remains rotated until the program commands **G69** or the program is reset. **G69** cancels all

coordinate rotation. To cancel only the last incremental rotation, command **G68** in **G91** mode with the opposite amount for **R**.

The **G68** or **G69** block does not move the nozzle. The *Absolute Position* window and System Variables indicate the nozzle position in the unrotated work coordinate system.

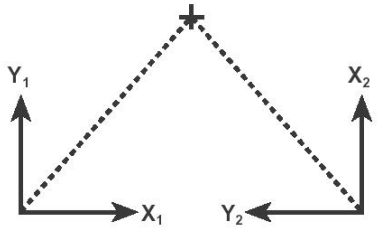
Example 1:

G68 X0 Y0 R30



Example 2:

G68 X5 Y5 R90



G90 AND G91 ABSOLUTE AND INCREMENTAL MODE

G90 Use Absolute mode

G91 Use Incremental mode

In **G90** Absolute mode, the nozzle moves to the coordinate location specified by the X and Y arguments in a **G00**, **G01**, **G02**, **G03,** or **G53** command. **G90** mode is active until the program commands **G91** mode. When each program starts, the default mode is **G90**.

In **G90** mode, X and Y coordinate values are modal. In other words, if a block does not specify X or Y positions, the control uses the last commanded value for X or Y.

In **G91** Incremental mode, the nozzle moves a distance from its starting location specified by X and Y in a **G00**, **G01**, **G02,** or **G03** command. **G91** mode is active until the program commands **G90** or the program ends. The control ignores **G53** commands while operating in **G91** mode.

G92 WORK COORDINATE SYSTEM SETTING

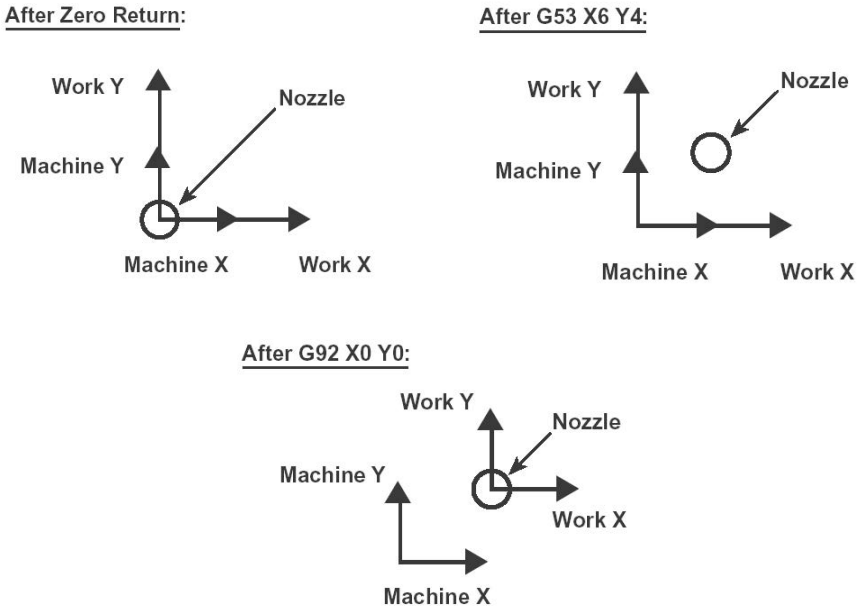
G92 X__ Y__ Set Work Coordinate System location

This command sets the work coordinate system location. When the machine completes the Axes Home operation, the control establishes the work coordinate system with X0, Y0 at Machine X0, Y0. The **G92** command can move the work coordinate system to any location. **G92** arguments X and Y define the new work coordinates corresponding to the nozzle position when the **G92** block is executed. The **G92** block does not move the nozzle.

The **G92 X0 Y0** command moves the work coordinate system X0 Y0 location to the current position of the nozzle. Programmers often use this command to begin a subprogram written in **G90** mode.

Example:

G92 X0 Y0



CUSTOM G CODES

The CINCINNATI BAAM System control has several built-in functions programmed with custom G Codes.

G22 TIP WIPE

G22 __ Tip Wipe

This command performs a tip wipe by calling the TipWipeMacro. The Z axis will move to the Clear height then the X and Y axes will then move to the Wipe location which should remove any plastic hanging from the extruder with the wiper. After this is finished, the X and Y axes will rapid back to their original location and then the Z axis will return to its initial position.

G22 DWELL WITH CONTINUE

G104 P__ Dwell with continue

The G104 is very similar to the G04 dwell with one difference. When a G104 is active, the CONTINUE button will flash for the duration of the G104. If the CONTINUE button is pressed while it is flashing, the dwell will be aborted and program execution will continue immediately.

M CODES

M codes are used for program control of specific machine functions, usually unrelated to X/Y-axis motion. See the **M Code List** for a table of all M codes supported by CINCINNATI Systems controls.

M00 PROGRAM STOP

M00 Program Stop

When the CNC executes a program block commanding **M00** (or **M0**), the program stops until the operator presses CYCLE START. **M00** places the CNC in a Cycle Stop condition. Modal information does not change.

M01 OPTIONAL STOP

M01 Program Stop (only if Optional Stop mode is active)

The **M01** (or **M1**) command has the same function as the **M00** command, except **M01** is only executed when Optional Stop Mode is active. Use the "Optional Stop" button on Program Modes Control Bar to enable/disable Optional Stop mode. **M01** is ignored if Optional Stop mode is disabled.

M02 END OF PROGRAM

M02 End of program

A CNC program can use **M02** (or **M2**) as the last block. This function disables all processing functions, resets all previously requested M codes and prevents further execution. The program does not rewind automatically after **M02** is executed. When the operator loads a program into the CNC, the control ignores any codes following **M02**.

M03 (P dwell) Extruder on CW

M03 (P dwell) Extruder on CW

This code turns the extruder on in the CW direction, which will cause the plastic to be extruded from the nozzle. The P argument is optional and can be used to dwell for the given time after the extruder is on.

M04 (P dwell)

M04 (P dwell)

This code turns the extruder on in the CCW direction (backwards). The P argument is optional and can be used to dwell after the extruder is turned on.

IMPORTANT: This M code should be used with caution! Instead of extruding material through the nozzle, this will cause plastic to be forced backwards through the feed system and can cause machine damage.

M05 Extruder Stop

M05 Extruder Stop

This code turns the extruder off.

M10 Dynamic Extruder Control ON

M10 Dynamic Extruder Control On

The code will turn on Dynamic Extruder Control. When this is on, the extruder speed will aotomatically be adjusted as the machine X and Y axes speed change. This helps prevent overfilling in corners and allows the feedrate override control to be used and not overfilling or underfilling the part.

M11 Dynamic Extruder Control OFF

M11 Dynamic Extruder Control Off

This code will turn off Dynamic Extruder Control. When this is off, the extruder speed is control exclusively with the S argument and the extruder overrides.

These codes set which path type is currently active. When a path type is active, the corresponding Path Type in the Override Section will highlight and that override will be active. For example, if the last code executed was a M14, the Infill override would be highlighted and the Infill override value would be in effect.

M12 Perimeter Path Type

M12 Perimeter Path Type

M13 Inset Path Type

M13 Inset Path Type

M14 Infill Path Type

M14 Infill Path Type

M15 Skin Path Type

M15 Skin Path Type

M16 No Path Type

M16 No Path type

These codes set which path type is currently active. When a path type is active, the corresponding Path Type in the Override Section will highlight and that override will be active. For example, if the last code executed was a M14, the Infill override would be highlighted and the Infill override value would be in effect.

M30 End of Program / Rewind

M30 End of program with rewind

Most CNC main programs use **M30** as the last block. This function disables all processing functions, resets the CNC, cancels all previously requested M codes, prevents further execution and rewinds the program.

M55 Tapper ON

M55 Tapper on

This code will turn on the tapper on the pellet feed system to help prevent clogging of the pellets. This is normally turned on whenever the extruder is turned on.

M56 Tapper OFF

M56 Tapper off

This will turn the tapper off. The tapper is usually turned off when the extruder is turned off.

M64 L speed Tamper ON

M64 L speed Tamper On

This code turns the material tamper on. The L argument is used to control the speed of the tamper. The L value can vary from 0.0 to 1.0 where 0 is 0% and 1.0 is 100%.

M65 Tamper OFF

M65 Tamper Off

This code turns the material tamper off.

M66 L accel Change X and Y accel rates

M66 L accel Change X and Y accel rates

This M code changes the accel / decel rates for the X and Y axes. The units of the L argument are in g's and it is limited to maximum accel rate in the machine configuration.

M68 Park

M68 Park

This code calls for the ParkMacro macro and moves the Z axis up to the clear position and then moves the X and Y axes to the Park position. These positions are set in the configuration.

M69 Purge Macro

M69 Purge Macro

F - Extruder rate (RPM) default 100

L - Return to location (0 = no, 1 = yes) Default 0

P - Purge time (seconds) default 60

S - Wait time after purge before wipe (seconds) Default 1

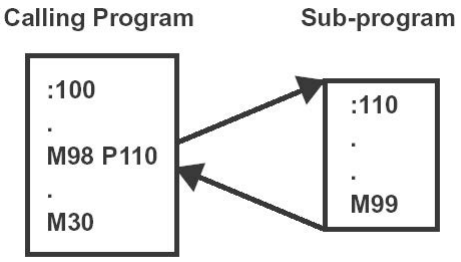
T - Wait time before purge (seconds) Default 1

This M code calls the PurgeMacro macro and is used to move the nozzle to the purge station and turn the extruder on for the specified time. The Continue button can be pressed during the purge time to abort the purge and continue with the program.

M98 SUBPROGRAM CALL

M98 P__ (L__) Subprogram call with no arguments

The **M98** command transfers control from the calling program to a subprogram. The **M98** block specifies the subprogram name after **P**. If the subprogram is in the same file as the calling program, only the program name is required. If the subprogram is in a separate file, **P** is followed by the file name including its extension (if any) and its path if different from the calling program.



The program can call the subprogram more than once by specifying the number of times with **L**.

Example: Calling a subprogram named "1200" three times:

M98 P1200 L3

When a program calls a subprogram with **M98**, the two programs share the same set of local variables (See the **SUBPROGRAMS AND MACROS** topic).

M99 END SUBPROGRAM

M99 (P__) End a subprogram and return control to the main program

The **M99** command returns control to the program that called the subprogram. The block following the subprogram call is executed next. Subprograms called with **M98** or **G65** can end with **M99.**

If a main program commands **M99**, the control restarts the main program from the beginning.

If the **M99** command includes the optional **P** argument, the subprogram returns to the calling program at the sequence number specified after **P**. If the **M99 P** command is in the main program, the control returns to the line number specified by **P** in the same program (same as **GOTO**).

Example:

M99 P500

(If commanded in a subprogram, this example returns to the calling program at line **N500**. If commanded in a main program, this example returns to line **N500** in the main program.)

PROGRAM STRUCTURE

NC Programs are instructions for motion interpolation, machine functions and program control. NC Programs for the CINCINNATI INCORPORATED System are ASCII text files written in the ISO format (International Standard Organization).

NC Programs execute from the program name down to the end of program statement. Each line is one NC block that may contain statements, expressions, program control, **G** code or **M** code commands. Each block can have a line number (also called the sequence number) assigned to it.

A program contains the following items:

- Program Name
- Program Body
- Optional Comments
- Program Body
- Optional Line Numbers
- Optional Block Delete Character (per line)
- End of Program (**M02** or **M30**), or End of Subprogram (**M99**)

PROGRAM NAME

Program names are required for subprograms included in the same file with other (main or sub) programs. Otherwise, program names are optional. The block containing the program name can begin with either a colon character (:) or the letter **O**, followed by a string of up to 128 alphanumeric characters (see Note). The end of the name is determined by a space.

***Note**: There is one case where the program name cannot include alphabetic characters: The program name must be an integer number (maximum 9 digits) when the program is a subprogram called by a macro and the macro call*

specifies the subprogram name as a macro argument.

PROGRAM BODY

The program body contains one or more NC blocks. When the program is displayed as a text file, each block is one line of text. At the end of every block is the ISO character for line feed.

When the operator loads a program, the control checks the syntax of each block. If a block contains a syntax error, the control displays the message: “Incorrect syntax for applications”.

A block of code may consist of any of the following:

- NC move command (**G** code)
- NC modal command (**G** code or **Feedrate**)
- Math function (variable assignment)
- Logic statement (**IF** [...] **THEN** ... , **GOTO** ... etc.)
- Machine (**M** code) function
- Macro call

Normally the control executes each block before proceeding to the next block. However, when a block assigns a math function to a variable, the control may look ahead and begin processing the math function during the execution of preceding blocks.

PROGRAM COMMENTS

Because NC program statements are usually not direct readable text, it is sometimes convenient to add comments to the program. For example, the top of a program may contain set-up instructions for the operator.

Comments can be added to the program by enclosing the comment characters between parentheses ( ), or by starting the comment with an apostrophe (‘).

When the control reads a left parenthesis in a program block, it interprets all characters to the right as a comment, until it finds a right parenthesis. The block terminates with the end of block character (line feed). When the control reads an apostrophe in a program block, it interprets all characters to the right as a comment, until it finds the end of block character.

Examples:

G92 X0 Y0 (SET WORK COORD.)

M98 P2000 ‘ CALL PROGRAM 2000

PROGRAM LINE NUMBERS

When a block directs execution to another block with a **GOTO** or **M99 P__** command, the destination block requires a line number. Otherwise, line numbers are optional. To assign a line number, begin the block with the character **N** followed by an integer. The line number has a usable range of 1 to 999999. The block may contain the block delete character before the line number.

Example:

N2300 G01 X50
(This example assigns line number 2300.)

The CNC Run Window displays line numbers with the program text.

BLOCK DELETE

The operator can control the execution of a block with the **Block Delete** function. When a block begins with the / (forward slash) character, the control does not execute the block if Block Delete program mode is active (see the **PROGRAM MODES** topic). The operator can toggle this function on or off any time during the execution of the NC program.

END OF PROGRAM

M02

M02 completes execution of a program. All previously requested M codes are reset and local variables are set to zero. However, the CNC Run Window does not return to the top of the program.

M99 (P_)

The **M99** block completes execution of a subprogram. The CNC Run Window returns to the program that called the subprogram and displays the next block.

M30

The **M30** block completes execution of a program. All previously requested **M** codes are reset, local variables are set to zero and the CNC Run Window returns to the top of the program.

SUBPROGRAMS AND MACROS

Subprograms are separate CNC programs that execute when called by another program. The subprogram returns to the calling program when finished. The control maintains modal conditions established in the calling program unless the subprogram changes the modal condition.

Subprograms are useful for repeating a series of commands. For example, a main program can use a subprogram to repeat a part on a sheet.

There are two types of subprogram calls, **M98 P_** and **G65 P_**. The call specifies the subprogram name after **P**. If the subprogram is in the same file as the calling program, then the call only needs the program name after **P**. If the subprogram is in a separate file, the call should command **P** followed by the file name including its extension (if any) and its path if different from the calling program. Each type of subprogram call has different properties and applications.

Examples:

M98 P__ (L_)

G65 P__ (L_) (A_ B_ C_ D_ etc. **)**

A single program block can call a subprogram more than once, by commanding **M98** or **G65** with **L** followed by the number of times to repeat the subprogram.

Example:

M98 P1000 (L3)

(This example calls a subprogram named “1000” three times before returning.)

LOCAL VARIABLES

The major difference between **M98** and **G65** is the treatment of local variables. When a program calls a subprogram with **M98**, the two programs share the same set of local variables (#1 through #99). In other words, “local” variables become “common” variables between a program and a subprogram called with **M98**. The control assigns undefined local variables a value of zero.

When a program calls a subprogram with **G65**, the subprogram has its own set of local variables. The calling program can assign values to most of the variables #1 through #26 in the subprogram by including arguments in the **G65** call. This allows the calling program to pass data to the subprogram without assigning separate common variables.

Each argument is a letter followed by a numerical value for its corresponding local variable. The calling program cannot use arguments **G**, **L**, **N**, **O,** or **P**. The subprogram can use local variables that would correspond to **G**, **L**, **N**, **O,** and **P**, but the calling program cannot assign their values with arguments in the **G65** call.

Since program names in the CINCINNATI control can have alphabetic characters, the **G65** block requires a space between the last character of the program name and the first argument letter.

Examples:

G65 PMYSUBA1
(This example calls a subprogram named “MYSUBA1”.)

G65 PMYSUB A1
(This example calls subprogram “MYSUB” with argument A equal to 1.)

To be consistent with local variable assignments used by other CNC controls, the CINCINNATI control assigns arguments **D** through **K** to local variables out of sequence with their alphabetical order.

This table shows how the control assigns local variables to **G65** arguments:

ARGUMENT	LOCAL VARIABLE
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

Example:

G65 P2000 X12.5 Y3.5
(This example calls a subprogram named "2000". When the program named "2000" starts, it has local variables #24 = 12.5 and #25 = 3.5.)

The format for subprograms is the same as other programs, except subprograms use **M99** instead of **M30** for the End of Program statement. If a subprogram does not end with **M99**, the control displays the message: "S*ubprogram call without a return statement"*.

NESTED SUB-PROGRAM CALLS

Subprograms can call other subprograms with **M98** or **G65**, until the total calls are nested 10 deep.

Note: *CINCINNATI Macros and Grid Macros are subprograms and contain subprogram calls. The total nesting limit available to the user is reduced when a program calls these macros.*

This table shows how CINCINNATI macro calls affect subprogram nesting:

MACRO	EQUIVALENT SUBPROGRAM CALLS
G73	3
G76	3
G83	3
G86	3
G88	3
G104	2
G65 P9800	2
G65 P9900	2

PROGRAM VARIABLES

LOCAL AND COMMON VARIABLES

Local and common variables can be displayed and edited by opening the *Local/Global Variables* window from *Variables* menu of the System HMI .

⚠ CAUTION

Editing variables while a program is running can cause improper execution and is strongly discouraged.

Note: *The **Local/Global Variables** window will display variables with the current values they have in the control buffer. However, the buffer may be several blocks ahead of the currently executing NC program block.*

Local Variables: #1 - #99

Each program has 99 local variables named **#1** through **#99**. Each subprogram called with **G65** also has its own set of local variables. Subprograms called with **M98** share the same set of local variables with the calling program. All local variables

are zero by default and return to zero after **M30** or Program Rewind.

Common Variables: #100 - #999

All programs and subprograms share a set of common variables named **#100** through **#999**. Common variables are not cleared by **M30**, pressing the "RESET" button or turning off control power.

SYSTEM VARIABLES

System variables give the programmer the ability to read and write information for special functions in the CNC.

There are four types of system variables:

SYSTEM VARIABLES	DESCRIPTION
#2000 - #2999	Offset Data
#3000 - #3999	CNC Data
#4000 - #4999	Modal Data
#5000 - #5999	Coordinate Data

Offset Data System Variables

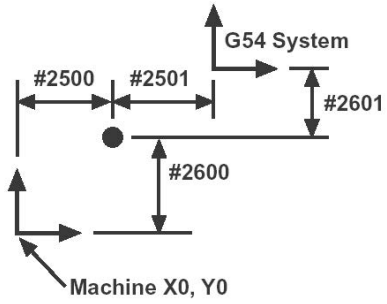
Work Coordinate System Offsets

An NC program can read the value of a Work Coordinate Offset by assigning the value of its system variable to a program variable, or by using the system variable in a math or logic statement. The program can also change the Work Offsets by assigning values to the system variables. The control interprets the variables in the active units (**G20** inch, **G21** mm).

SYSTEM VARIABLE	Work Offset Group	Axis
#2500	External	X
#2600	External	Y
#2501	1 (G54)	X
#2601	1 (G54)	Y
#2502	2 (G55)	X
#2602	2 (G55)	Y
#2503	3 (G56)	X
#2603	3 (G56)	Y
#2504	4 (G57)	X
#2604	4 (G57)	Y
#2505	5 (G58)	X
#2605	5 (G58)	Y
#2506	6 (G59)	X
#2606	6 (G59)	Y

External work offsets **(#2500** and **#2600)** are normally zero. Programs use **#2500** and **#2600** to change the distance from machine **X**0, **Y**0 to a common reference point for the **G54** through **G59** work coordinate systems.

Example:



Modal Data System Variables

Programs can use the **#4000** series system variables in a math or logic statement. A program cannot change a **#4000** series variable by assigning a value to the system variable.

SYSTEM VARIABLE	DESCRIPTION
#4001	0, 1, 2, or 3 for G0, G1, G2 or G3
#4003	90 or 91 for G90 or G91
#4006	20 or 21 for G20 or G21
#4007	40, 41 or 42 for G40, G41 or G42
#4008	54 through 59 for G54 through G59
#4011	50 or 51 for G50 or G51
#4015	61 or 64 for G61 or G64
#4016	68 or 69 for G68 or G69
#4109	Modal Feedrate for G1, G2 and G3
#4018	Dynamic Extruder Mode

Coordinate System Variables

A program can determine the machine or work coordinates of the last completed block by reading system variables. The control maintains the variables in the same units as the active units mode (**G20** inch, **G21** mm). The program cannot change these system variables by assigning a value to the variable.

Machine Coordinates:

#5021 Machine x position

#5022 Machine y position

#5023 Machine z position

#5061 Position Capture machine x position

#5062 Position Capture machine y position

Work Coordinates:

#5041 Work x position

#5042 Work y position

#5043 Work z position

AUXILIARY FUNCTIONS

MATH FUNCTIONS

NC programs can use the math functions listed in the following table to assign a calculated value to a variable, or to substitute a calculation for a numerical value. When processing program blocks containing math expressions, the CNC applies the following default order of precedence to the math operations:

1. Functions (SQRT, ABS, TAN, etc.)
2. Multiplication and division
3. Addition and subtraction

In the following examples, “a” represents a variable and “b” and “c” represent variables, constants or functions.

FUNCTION	EXAMPLE
Assignment	a = b
Addition	a = b + c
Subtraction	a = b - c
Multiplication	a = b * c
Division	a = b / c
Binary Addition	a = b OR c
Binary Subtraction	a = b XOR c
Binary Multiplication	a = b AND c
Sine	a = SIN [b]
Cosine	a = COS [b]
Tangent	a = TAN [b]
Arc Tangent	a = ATAN [b]
Square Root	a = SQRT [b]
Rounding	a = ROUND [b]
Truncating	a = FIX [b]
Add 1 for fraction	a = FUP [b]
Absolute Value	a = ABS [b]
BCD to Binary	a = BIN [b]
Binary to BCD	a = BCD [b]

SIN [ ] Sine of an angle in degrees.

COS [ ] Cosine of an angle in degrees.

TAN [ ] Tangent of an angle in degrees.

ATAN [ ] Arc-tangent of an expression.
The result of the ATAN function is in degrees, between -90 and +90.

SQRT [ ] Square root of a positive number.
Evaluating the square root of a negative number produces an error message.

ROUND [ ] Rounding off a number.
Expressions with fractional portions 0.5 and above are rounded to the next higher integer. Expressions with fractional portions less than 0.5 are rounded to their integer value.

FIX [ ] Truncating a number.
Any fractional portion is ignored. The number is reduced to its integer value only.

FUP [ ] Add 1 for fraction.
If the number has any fractional portion, the FUP function removes the fraction and adds 1 to the integer portion. If the number is already an integer (no fractional part), it stays the same.

ABS [ ] Absolute Value
ABS [ ] returns the absolute value of a variable or function.

BIN [ ] BCD to Binary number conversion
The BIN function converts the specified value from Binary Coded Decimal to Binary.

BCD [ ] Binary to BCD number conversion
The BCD function converts the specified value from Binary to Binary Coded Decimal.

Braces [ ]

Math expressions can use braces to override the default order of precedence for math operators, and identify conditional expressions. There is no limit on the number of nested braces.

Example 1:

#1 = 3 * [2 + 3]

The braces set the priority of the math functions; addition is performed before multiplication in Example 1.

Example 2:

#2 = SIN[2 * [3 *[2 + 3]]]

In Example 2, the statement completes addition first. The sum is multiplied by 3 and that product is multiplied by 2. The SIN function operates on the product of the multiplication.

Each left brace must have a right brace. If the statement contains an error in brace syntax, the control displays the message "*Brace open/close error"*.

LOGIC FUNCTIONS

Logic functions include conditional expressions and program control commands.

Conditional Expressions

A program uses a conditional expression to compare the value of a variable, constant or calculation with another variable, constant or calculation. The program block can use the result of the comparison to direct program flow.

COMPARISON	EXAMPLE
Equal	[b EQ c]
Not Equal	[b NE c]
Greater Than	[b GT c]
Less Than	[b LT c]
Greater Than or Equal	[b GE c]
Less Than or Equal	[b LE c]

The CNC evaluates the conditional expression for proper syntax and a true or false condition. The evaluation of a conditional expression occurs from left to right.

Program Control Commands

A program can determine which blocks the control executes by using the program control commands:

- GOTO
- IF [ ] GOTO
- IF [ ] THEN . . ELSE . . ENDIF
- WHILE [ ] . . END

GOTO Statement

The control normally executes program blocks in sequential order (top to bottom through the program). A program can direct the control to execute any numbered block with the **GOTO nnn** command, where "nnn" is the sequence number of the destination block.

A program block specifies its sequence number with '**N**' followed by an integer number. To direct the control to a block with a sequence number, command **GOTO** followed by the numerical value of the sequence number. The **GOTO** command does not include the "**N**" character.

Example:

```
N020 GOTO 50
N030 . . .
N040 . . .
N050 G01 X10 Y5
```

*(In this example, the control executes the block with sequence number **N050** immediately after the block with sequence number **N020**.)*

IF [ ] GOTO Statement

IF [<conditional expression>] **GOTO** <line>

If the expression is true, the program jumps to the specified line number. If false, the program continues with the next block.

IF [ ] THEN … ENDIF Statement

IF [<conditional expression>]
THEN < expression >
ENDIF

The CNC evaluates the conditional expression. If true, the control executes the block containing **THEN**. If the expression is false, the program jumps to the **ENDIF** block.

The **THEN** block must be a separate block immediately following the **IF** block. Each **IF THEN** pair must be followed by a separate **ENDIF** block. **IF THEN** statements can be nested four deep.

Example:

```
IF [#1 EQ #2]
THEN #3 = #4 / #5
ENDIF
```

IF [ ] THEN … ELSE … ENDIF Statement

IF [<conditional expression>**]**
THEN <expression>
ELSE <expression>
ENDIF

If the conditional expression is true, the control executes the block containing **THEN** and the program jumps to the **ENDIF** block. If the expression is false, the control executes the block containing **ELSE**.

The **THEN** block must be a separate block immediately following the **IF** block. The **ELSE** and **ENDIF** blocks must also be separate blocks. **IF..THEN..ELSE..ENDIF** statements can be nested four deep.

WHILE Statements

The control accepts two types of **WHILE** statements:

WHILE . . **END** Statement:

WHILE [<conditional expression>**] DO m**
.
.
ENDm

(In the **WHILE** . . **END** structure, "**m**" is an integer from 1 to 3, used to identify nested loops.)

WHILE . . ENDWHILE Statement:

WHILE [<conditional expression>]
.
.
ENDWHILE (one word)

The control evaluates the conditional expression. If true, the control executes the block(s) between **WHILE** and **END** (or **ENDWHILE**), then the program returns to the **WHILE** block and repeats the process. When the expression becomes false, the program jumps to the block after **END** (or **ENDWHILE**).

WHILE loops can be nested up to five deep. To avoid the possibility of an infinite loop, a block between **WHILE** and **END** normally changes the status of the conditional expression.

Example:

```
#1=0
WHILE [#1 LE 50] DO2
.
.
#1=#1+1
END2
```

The blocks between **WHILE** and **END** may include a **GOTO** command to exit the loop.

NC CODE LIST

M CODE LIST		
M0	Program Stop	Cycle operation is stopped. All existing modal information remains unchanged. Cycle operation is resumed by depressing “CYCLE START”.
M1	Optional Stop	Same as M00 except code is ignored unless Optional Stop Mode is active.
M2	End of Program	End of main program. Click “REWIND” or load a new program to continue.
M3	Extruder CW (normal operation)	
M4	Extruder CCW (backwards)	
M5	Extruder Stop	
M10	Turn ON Dynamic Extruder Control	
M11	Turn OFF Dynamic Extruder Control	
M12	Perimeter Override	
M13	Inset Override Active	
M14	Infill Override Active	
M15	Skin Override Active	
M16	No Path Type Override	
M30	End of program / Rewind	End of main program with rewind. Press “CYCLE START” to repeat entire program.
M55	Clog Prevention Tap Solenoid ON	
M56	Clog Prevention Tap Solenoid OFF	
M64	Tamper ON	
M65	Tamper OFF	
M66	Change Gantry XY Accel Limits (L-accel limit in g’s)	
M68	Park Macro	
M69	Purge Macro	F - Extruder rate (RPM) default 100 L - Return to location (0 = no, 1 = yes) Default 0 P - Purge time (seconds) default 60 S - Wait time after purge before wipe (seconds) Default 1 T - Wait time before purge (seconds) Default 1
M98	Subprogram call	This code is used to call a subprogram.
M99	End of Subprogram	This code indicates the end of a subprogram and returns control to the main program.

G CODE LIST

G CODE LIST	
G00	Positioning-Rapid Traverse X/Y axes are commanded to move at top speed to the end point as specified by G0X<value>Y<value>. NOTE: Do not use G0 for ...
G01	Linear Interpolation X/Y axes are commanded to move at programmed feedrate to the end point as specified by G01X<value>Y<value>.
G02	Circular Interpolation CW X/Y axes are commanded to move at programmed feedrate, in a clockwise arc, to the end point as specified by G02X<value>Y<value>i<value>j<value>.
G03	Circular Interpolation CCW X/Y axes are commanded to move at programmed feedrate, in a counterclockwise arc, to the end point as specified by G02X<value>Y<value>i<value>j<value>.
G04	Dwell X/Y axes are commanded to a stop for a specified time duration. G04P<value> where <value> is expressed in milliseconds.
G09	Exact Stop X/Y axes are commanded to a brief stop at end point. For rectangular moves, this will ensure a sharp corner.
G20	Inch mode Inch mode is specified. All arguments, including X/Y arguments, are expressed in inches.
G21	Metric mode Metric mode is specified. All arguments, including X/Y arguments, are expressed in millimeters.
G50	Scaling Cancel Scaling mode is canceled.
G51	Scaling Scaling mode is enabled.
G52	Local Coordinate System Setting
G53	Machine Coordinate System Selection X/Y axes are commanded to move to the ACTUAL machine position specified in X<value>Y<value>.
G54	Work Coordinate System 1 Selection Work coordinate system #1 is specified.
G55	Work Coordinate System 2 Selection Work coordinate system #2 is specified.
G56	Work Coordinate System 3 Selection Work coordinate system #3 is specified.
G57	Work Coordinate System 4 Selection Work coordinate system #4 is specified.
G58	Work Coordinate System 5 Selection Work coordinate system #5 is specified.
G59	Work Coordinate System 6 Selection Work coordinate system #6 is specified.
G61	Exact Stop Mode All subsequent X/Y axes are commanded to a brief stop at end point. For rectangular moves, this will ensure a sharp corner.
G64	??? Mode (Cancel G61 and G62)
G65	Macro Call
G68	Coordinate System Rotation Coordinate system rotation is enabled.
G69	Coordinate System Rotation Cancel
G90	Absolute Command All subsequent X/Y arguments will be interpreted as ACTUAL desired positions / endpoints in the active coordinate system.
G91	Incremental Command All subsequent X/Y arguments will be interpreted as RELATIVE to the present position.
G92	Work Coordinates Change

CINCINNATI®