



G Code Support

G-Codes

Command	Description	Support	Comment
G00	Rapid positioning. (Max speed for each axis)	Yes	G0 moves use MachineData.Prms.MaxDirectVelocity, MachineData.Prms.DirectAcceleration, and MachineData.Prms.DirectDeceleration.
G01	Linear interpolation	Yes	G1 G2 G3 moves reference the F register for feedrate. If the F register is zero, then MachineData.Prms.MaxVelocity is applied. MachineData.Prms.Acceleration and MachineData.Prms.Deceleration are also referenced.
G02	CW circular interpolation	Yes	Use R for radius mode or I,J,K for circle center.
G03	CCW circular interpolation	Yes	Use R for radius mode or I,J,K for circle center.
G04	Dwell	Yes	Provide the dwell time as milliseconds with the P parameter or as seconds with X.
G05	High-precision contour control (HPCC).	No	Uses a deep look-ahead buffer and simulation processing to provide better axis movement acceleration and deceleration during contour milling.
G06	Non uniform rational B Spline machining.	No	Uses a deep look-ahead buffer and simulation processing to provide better axis movement acceleration and deceleration during contour milling.
G07	Imaginary axis designation	No	Uses a deep look-ahead buffer and simulation processing to provide better axis movement acceleration and deceleration during contour milling.
G09	Exact Stop, non-modal.	Yes	Forces the move to come to a complete stop rather than blending with the next move if there is another motion command following. Modal version is G61.
G10	Select work coordinate and tool offsets.	No	Not supported.
G11	Data write cancel	No	
G12	Full-circle clockwise interpolation	No	
G13	Full-circle counterclockwise interpolation	No	
G17	XY Plane Selection	No	This is the default.
G18	ZX Plane Selection	No	
G19	YZ Plane Selection	No	

Command	Description	Support	Comment
G20	Programming in inches	Yes	The Group Toolbox G-Code feature will default to the user units configured in the Hardware Configuration. If all G-Code files will contain position data in the configured user units of the machine, G20 / G21 are not required. If G-Code files may contain different user units, the application program must read parameter 1813 using the HC_ReadParameter function block to obtain the configured user units. Copy this parameter value into PathData.HC_UserUnits. This allows the G-Code parser to convert positions in the G-Code file to the configured units of the machine.
G21	Programming in millimeters	Yes	
G28	Return to the Home Position	Yes	Return to the machine's reference or Origin position. Specify this position in the MachineStruct.Origin[]. At least one of X, Y, or Z axes must be specified with G28 to indicate a via point to pass through before moving to the Origin.
G30	Return to the Secondary Home Position	No	Takes a P address specifying which machine zero point is desired, if the machine has several secondary points (P1 to P4).
G31	Skip Function	No	Used for probes and tool length measurement systems.
G32	Single-point threading, longhand style	No	Similar to G01 linear interpolation, except with automatic spindle synchronization for single-point threading.
G33	Constant-pitch threading	No	
G34	Variable-pitch threading	No	
G40	Tool radius compensation off	Yes	Turns off cutter radius compensation. Cancels G41 or G42.
G41	Tool radius compensation left	Yes	Creates a left tool compensation along the XY plane. Provide Tool Data via the ToolStruct connected to Read_GCode_File or Read_GCode_Stream.
G42	Tool radius compensation right	Yes	Creates a right tool compensation along the XY plane. Provide Tool Data via the ToolStruct connected to Read_GCode_File or Read_GCode_Stream.
G43	Tool height offset compensation negative	Yes	Uses the H or T register as the tool length offset. Provide Tool Data via the ToolStruct connected to Read_GCode_File or Read_GCode_Stream. Only supported when using MachineData.Emulation = GTB_Emulation#mode2.
G44	Tool height offset compensation positive	No	
G45	Axis offset single increase	No	
G46	Axis offset single decrease	No	
G47	Axis offset double increase	No	
G48	Axis offset double decrease	No	
G49	Tool length offset compensation cancel	Yes	Only supported when using MachineData.Emulation = GTB_Emulation#mode2.
G50	Define the maximum spindle speed/Scaling function cancel	No	
G52	Local Coordinate System. This is an offset from the current offset	No	
G53	Machine coordinate system. Takes absolute coordinates (X,Y,Z,A,B,C) with reference to machine zero rather than program zero.	Yes	Make a move in the Machine Coordinate System (MCS.) NOTE: Behavior changed for Group Toolbox v361 to comply with traditional (non modal) G-Code behavior. Prior to v361, executing a G53 would not initiate any motion by itself.
G54	Work Coordinate System 1	Yes	Selects the offsets in MachineStruct.CoordinateSystem.Offset [1] to be used for moves such as G0, G1, G2, G3.
G55	Work Coordinate System 2	Yes	Selects the offsets in MachineStruct.CoordinateSystem.Offset [2] to be used for moves such as G0, G1, G2, G3.

Command	Description	Support	Comment
G56	Work Coordinate System 3	Yes	Selects the offsets in MachineStruct.CoordinateSystem.Offset [3] to be used for moves such as G0, G1, G2, G3.
G57	Work Coordinate System 4	Yes	Selects the offsets in MachineStruct.CoordinateSystem.Offset [4] to be used for moves such as G0, G1, G2, G3.
G58	Work Coordinate System 5	Yes	Selects the offsets in MachineStruct.CoordinateSystem.Offset [5] to be used for moves such as G0, G1, G2, G3.
G59	Work Coordinate System 6	Yes	Selects the offsets in MachineStruct.CoordinateSystem.Offset [6] to be used for moves such as G0, G1, G2, G3. G59.1 through G59.3 are also supported.
G61	Exact stop check, modal. Can be canceled with G64. Non-modal version is G09.	Yes	
G62	Automatic Corner Override	No	
G64	Default cutting mode. Cancels G61	Yes	
G70	Fixed cycle, multiple repetitive cycle, for finishing (including contours)	No	
G71	Fixed cycle, multiple repetitive cycle, for roughing (Z-axis emphasis)	No	
G72	Fixed cycle, multiple repetitive cycle, for roughing (X-axis emphasis)	No	
G73	Fixed cycle, multiple repetitive cycle, for roughing, with pattern repetition	No	
G74	Tapping cycle for milling, left hand thread, M04 spindle direction	No	
G75	Peck grooving cycle for turning	No	
G76	Fine boring cycle for milling	No	
G78	Tangent Motion Enable	Yes	Synchronizes a theta axis (which is not part of the AxesGroup) to the XY plane of a path. The external axis must be defined as shown in the example for MC_MovePath . Requires firmware 3.3.0 or higher.
G79	Tangent Motion Disable	Yes	
G80	Simple Drilling Cycle	No	
G81	Drilling Cycle with Dwell	No	
G82	Spot Drilling cycle (full retraction from pecks)	No	
G83	Peck Drilling cycle (full retraction from pecks)	No	
G84	Tapping cycle, right hand thread, M03 spindle direction	No	
G90	Absolute Positioning	Yes	
G90	Lathe: Straight Cutting Cycle	Yes	If MachineStruct.MachineType is set to Lathe, G90 commands will use a Straight Cutting Cycle macro.
G91	Incremental Positioning	Yes	
G92	Set Position	Yes	Starting in v361, all axes are supported. G92 with no axes specified clears any existing offsets. These "LocalZeroOffsets" are applied to all Work offsets G54 ~ G59 in addition to any offsets established for G54 ~ G59 in MachineData.CoordinateSystem []. The redefined positions (stored in MachineData.LocalZero

Command	Description	Support	Comment
			oOffsets[]) are temporary, meaning they are lost after power cycle or PLC restart, unless the MachineData variable is set as 'retain' data in the MotionWorks IEC Global variables list.
G93	Inverse Time Mode	Yes	Support added for Group Toolbox v350 release.
G94	Units per Minute Feed Rate Mode	Yes	This was always the default feedrate mode, but the G94 command was officially recognized as of the v350 release.
G96	Constant surface speed (CSS)	No	Varies spindle speed automatically to achieve a constant surface speed.
G97	Constant spindle speed	Yes	Takes an S address integer, which is interpreted as RPM. The Group Toolbox does not have any built in support for spindles, other than datatypes to hold information for operating a spindle by customizing the main project using this toolbox. See MachineStruct.Spindle[] .
G98	Absolute Programming	Yes	Only valid if MachineStruct.Emulation:=GTB_Emulation#Mode1
G99	Incremental Programming	Yes	Only valid if MachineStruct.Emulation:=GTB_Emulation#Mode1

M Codes

Command	Description	Support	Comment
M00	Non-optional Stop. Machine always stops here.	Yes	Requires Group Toolbox v352 or higher.
M01	Optional Stop. Only stops if Optional stop flag is set. See MachineStruct.Command.ProgramStop	Yes	Requires Group Toolbox v352 or higher.
M02	End of Program	Yes	
M03	Spindle On (clockwise rotation)	Yes	Supported as MC_MovePath.OutputFlag.X3. The Group Toolbox does not have any built in support for spindles, other than datatypes to hold information for operating a spindle by customizing the main project using this toolbox. See MachineStruct.Spindle[] . For multi spindle support, use commands M143, M144, M145 and M153, M154, M155.
M04	Spindle On (counterclockwise rotation)	Yes	Supported as MC_MovePath.OutputFlag.X4
M05	Spindle Stop	Yes	Supported as MC_MovePath.OutputFlag.X3 and X4
M06	Automatic Tool Change	No	
M07	Coolant On (mist)	Yes	Supported as MC_MovePath.OutputFlag.X0
M08	Coolant On (Flood)	Yes	Supported as MC_MovePath.OutputFlag.X1
M09	Coolant Off	Yes	Supported as MC_MovePath.OutputFlag.X0 and X1
M10	Pallet Clamp On	Yes	Supported as MC_MovePath.OutputFlag.X2
M11	Pallet Clamp Off	Yes	Supported as MC_MovePath.OutputFlag.X2
M13	Spindle on (clockwise rotation) and coolant on (flood)	Yes	Supported as MC_MovePath.OutputFlags X1 and X3
M19	Spindle Orientation	No	
M21	Mirror, X-Axis	No	
M22	Mirror, Y-Axis	No	
M23	Mirror Off	No	
M24	Thread gradual pullout off	No	
M26	Axis Clamping	No	
M27	Axis Clamping	No	

Command	Description	Support	Comment
M30	End of program	Yes	
M41	Gear select 1	No	
M42	Gear select 2	No	
M43	Gear select 3	No	
M44	Gear select 4	No	
M48	Feedrate override allowed	No	
M49	Feedrate override NOT allowed	No	
M52	Unload last tool from spindle	No	
M62	Set Digital Output On	Yes	The P parameter specifies the digital output number 1-32 (Bit of MC_MovePath.OutputFlags)
M63	Set Digital Output Off	Yes	The P parameter specifies the digital output number 1-32 (Bit of MC_MovePath.OutputFlags)
M60	Automatic Pallet change (APC)	No	
M66	Wait for Input	Yes	Reference http://linux-cnc.org/docs/html/gcode/m-code.html#mcode:m66 . P parameter: 0 ~ 31 is mapped to MC_MovePath.InputFlags. P1 is bit 0, P2 is bit 1, and so on. E parameter: Not supported. L Parameter: Only Mode 4 is supported; MC_MovePath simply holds up sequencing until the required input conditions (bit flags) are met. If other input states are desired, connect the necessary logic to MC_MovePath.InputFlags. Q parameter: timeout in seconds.
M80	Custom: Cancel BreakOut and Head.	Yes	Cancel BreakOut and Head.
M81	Custom: Set Head.	Yes	Set Head (Mapped to MC_MovePath.OutputFlags.X5)
M82	Custom: Set BreakOut.	Yes	Set BreakOut (Mapped to MC_MovePath.OutputFlags.X6)
M98	Subprogram call	Yes	16 sub programs supported. Sub program must be within the same file as the main program. (MPiec G-Code feature only supports one file at a time.)
M99	Subprogram end	Yes	
M104	Set Extruder Temperature	Yes	http://reprap.org/wiki/G-code#M104:_Set_Extruder_Temperature
M106	Set Fan On (and speed)	Yes	http://reprap.org/wiki/G-code#M106:_Fan_On
M107	Set Fan Off	Yes	http://reprap.org/wiki/G-code#M107:_Fan_Off
M109	Set Extruder Temperature and Wait	Yes	https://reprap.org/wiki/G-code#M109:_Set_Extruder_Temperature_and_Wait
M140	Set Bed Temperature	Yes	https://reprap.org/wiki/G-code#M140:_Set_Bed_Temperature_.28Fast.29
M190	Set Bed Temperature and Wait	Yes	https://reprap.org/wiki/G-code#M190:_Wait_for_bed_temperature_to_reach_target_temp
M143	Spindle [1] On (clockwise rotation)	Yes	The Group Toolbox does not have any built in support for spindles, other than datatypes to hold information for operating a spindle by customizing the main project using this toolbox. See MachineStruct.Spindle[] .
M144	Spindle [1] On (counterclockwise rotation)	Yes	
M145	Spindle [1] Stop	Yes	
M153	Spindle [2] On (clockwise rotation)	Yes	
M154	Spindle [2] On (counterclockwise rotation)	Yes	
M155	Spindle [2] Stop	Yes	
M207	Retract filament	Yes	http://reprap.org/wiki/G-code#M207:_Set_retract_length

Parameters

Command	Description	Support	Comment
A	Absolute or incremental position of A axis (rotational axis around X axis)	No	This is a Rotational position. (Rx) Actual implementation handled in firmware based on Hardware Configuration support for selected mechanisms, or must implement custom kinematics in the application layer of the MotionWorks IEC project.
B	Absolute or incremental position of B axis (rotational axis around Y axis)	No	This is a Rotational position. (Ry) Actual implementation handled in firmware based on Hardware Configuration support for selected mechanisms, or must implement custom kinematics in the application layer of the MotionWorks IEC project.
C	Absolute or incremental position of C axis (rotational axis around Z axis)	Yes	This is a Rotational position. (Rz) Actual implementation handled in firmware based on Hardware Configuration support for selected mechanisms.
D	Defines diameter or radial offset used for cutter compensation. D is used for depth of cut on lathes.	No	
E	3D Printer - Extruder position	Yes	MachineStruct.MachineType must be set to GTB_MachineType#Printer. Lathe mode not supported for precision feedrate for threading
E	Mode1 emulation: Set acceleration in units/sec.	Yes	If MachineStruct.Emulation:=GTB_Emulation#Mode1 then the E register is used as acceleration in units/sec ² . This emulation mode is mutually exclusive with MachineType GTB_MachineType#Printer.
F	Feed rate	Yes	Specify in units / minute. If MachineStruct.Emulation:=GTB_Emulation#Mode1 then the F register is interpreted as feedrate in units/sec.
H	Tool length offset	Yes	Same as T register.
I	Arc Center in X axis for G02 or G03	Yes	This is the relative X distance to the center of the circle from the beginning of the arc.
J	Arc Center in Y axis for G02 or G03	Yes	This is the relative Y distance to the center of the circle from the beginning of the arc.
K	Arc Center in Z axis for G02 or G03	Yes	This is the relative Z distance to the center of the circle from the beginning of the arc.
L	Fixed cycle loop count	Yes	Use with M98.
N	Line number (optional)	Yes	If provided, this information is copied to PathData.Segment [].Label and is useful for troubleshooting.
O	Program Name	Yes	Use with M98.
P	Dwell time in milliseconds for G04, and parameter used by some canned cycles, and for jumps.	Yes	This parameter has multiple functions based on the G-Code.
Q	Peck increment in canned cycles (G73 and G83)	No	
R	Radius of an arc	Yes	For use with G02 and G03.
S	Defines speed, either spindle RPM or surface speed depending on the mode. (G96 or G97)	Yes	
T	Tool Selection	Yes	For 3P Printers, the single command on a line T0, T1, or T2 will switch the active Extruder. A G1 instruction is auto inserted to move the selected Extruder head the offset specified in MachineData.Printer.Extruder[[]].

Command	Description	Support	Comment
U	Incremental X axis (Ignores G90 and G91)	No	
V	Incremental Y axis (Ignores G90 and G91)	No	
W	Incremental W axis (Ignores G90 and G91)	No	
X	X axis position	Yes	This is a Cartesian position within the working space of the mechanism. Actual implementation handled in firmware based on Hardware Configuration support for selected mechanisms.
Y	Y axis position	Yes	This is a Cartesian position within the working space of the mechanism. Actual implementation handled in firmware based on Hardware Configuration support for selected mechanisms.
Z	Z axis position	Yes	This is a Cartesian position within the working space of the mechanism. Actual implementation handled in firmware based on Hardware Configuration support for selected mechanisms.

Other Supported Commands

Variable and basic logical commands are supported with the following limitations:

- When using logical statements (M98 or IF command), the entire G-Code file contents must be loaded into MyPath.Segment[]. The MyPath.Buffer.Overwritten flag must not have been set TRUE by the G-Code Processor. This is because the function blocks can only reference data already processed and in the PLC memory area; they cannot search for data in the G-Code file on the controller flash file system, and if the file is being streamed, they have no way to randomly access data which may already be overwritten by the circular buffer, or may not have been sent to the MPiec controller yet.
- Only simple assignments are supported. No nested parenthesis or math equations involving more than one operator are supported.
- Nested subroutines can be called with a maximum depth of 16, otherwise MC_MovePath will output the ErrorID 10626 - Stack Overflow.
- Up to 255 variables assignments or expressions can exist in one G-Code file.

Logic / Expression support

Operator	Example	Comment
IF	IF [expression] 'N' line number	Must include square brackets, parenthesis not allowed. If the logical expression is true, then the line number specified by the N parameter will be executed next. The G-Code file must contain N line numbers for the IF command to work.
LT or <	Less than	IF [#5 < #18] N32
LE or <=	Less than or equal	IF [#362 LE 27] N458
NE or <>	Not equal	IF [#1 <> #2] N3
EQ or =	Equal	IF [#1 = 0] N27
GT or >	Greater than	IF [#500 GT #501] N123
GE or >=	Greater than or equal	IF [#5 >= 18.5] N21

The keywords THEN ELSE END_IF and GOTO are not supported.

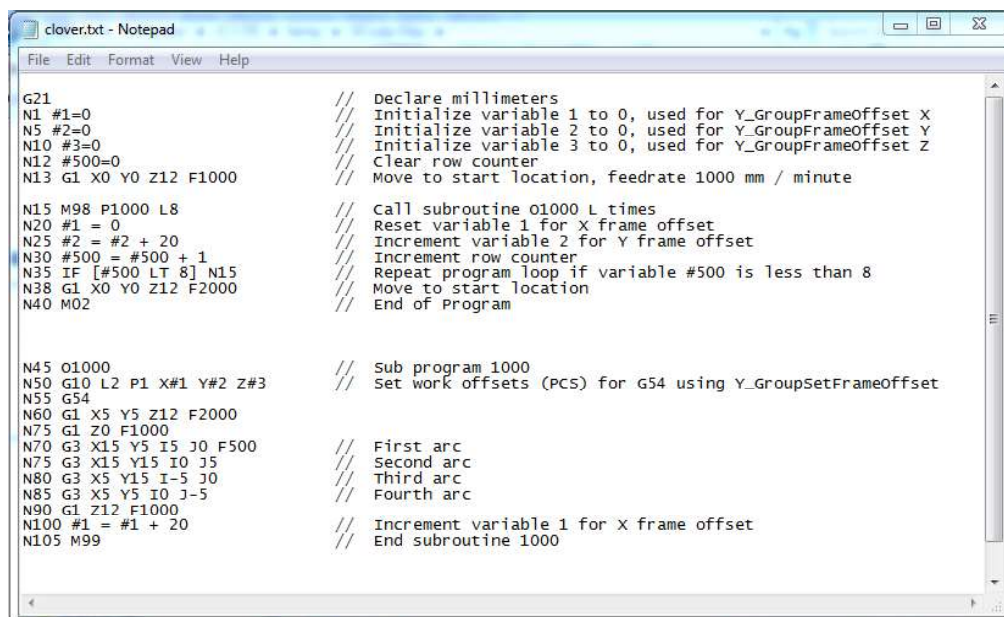
Other Keyword support

Character	Example	Comment
/	/G1 X6 Y17	This is an optional block skip command. Lines with a single leading '/' character will be processed only if the MachineStruct.Control.BlockSkip flag is set. Set this flag by customizing the MotionWorks IEC application or GUI software as necessary. Double slashes are always recognized as the start of a comment.

Subroutine and Variable Example:

The following example shows how variables can be assigned and subroutines can be called. This program creates a clover pattern, and repeats the shape in an 8 by 8 pattern by changing the Part Coordinate System using looping and variables.

[Link to video](#)



```
clover.txt - Notepad
File Edit Format View Help

G21                                // Declare millimeters
N1 #1=0                            // Initialize variable 1 to 0, used for Y_GroupFrameOffset X
N5 #2=0                            // Initialize variable 2 to 0, used for Y_GroupFrameOffset Y
N10 #3=0                           // Initialize variable 3 to 0, used for Y_GroupFrameOffset Z
N12 #500=0                         // Clear row counter
N13 G1 X0 Y0 Z12 F1000            // Move to start location, feedrate 1000 mm / minute

N15 M98 P1000 L8                  // Call subroutine 01000 L times
N20 #1 = 0                        // Reset variable 1 for X frame offset
N25 #2 = #2 + 20                  // Increment variable 2 for Y frame offset
N30 #500 = #500 + 1               // Increment row counter
N35 IF [#500 LT 8] N15            // Repeat program loop if variable #500 is less than 8
N38 G1 X0 Y0 Z12 F2000           // Move to start location
N40 M02                           // End of Program

N45 01000                         // Sub program 1000
N50 G10 L2 P1 X#1 Y#2 Z#3        // Set work offsets (PCS) for G54 using Y_GroupSetFrameOffset
N55 G54
N60 G1 X5 Y5 Z12 F2000
N75 G1 Z0 F1000
N70 G3 X15 Y5 I5 J0 F500         // First arc
N75 G3 X15 Y15 I0 J5             // Second arc
N80 G3 X5 Y15 I-5 J0             // Third arc
N85 G3 X5 Y5 I0 J-5              // Fourth arc
N90 G1 Z12 F1000
N100 #1 = #1 + 20                 // Increment variable 1 for x frame offset
N105 M99                         // End subroutine 1000
```




G-Code Emulation Modes

The default is no emulation mode selected (MyMachine.Emulation:=GTB_Emulation#na). Only a single mode can be selected. Emulation modes alter the behavior of G-Code solution in the following ways:

Mode1

Feature	Behavior
E Register	Interpret E register as positive values for acceleration and negative values for deceleration in units/sec ² .
F Register	Velocity is units/sec rather than units/min.

Mode2

Feature	Behavior
F	<p>Feedrates for XY, Z, and rotational motion can be specified separately using axes letters as parameters. For example:</p> <p>F 300 XY</p> <p>F 50 Z</p> <p>Feedrates will be checked such that the limit for a each axis is not exceeded. For example if the Z feedrate has been set at 50 units/min, and a command such as G1 X40 Y200 Z12 F600 is executed, the overall velocity of the command may be reduced to less than 600 if the Z component of the move would require it to exceed 50.</p>
G0	Rapid moves unaffected by Feedrate Override.
G92	<p>If executed by itself, clears all MachineData.LocalZeroOffset[] to zero.</p> <p>If Axes registers are provided, sets the new LocalZeroOffset for the specified axes.</p>
G53	If executed by with no axes specified, the Z axis will be moved to MachineData.Origin[3] .
H	Home Preset commands can be used with G1. For example G1 H#3 will move the machine to the pre determined location identified as Home #3.
L30 / L31	Commands are recognized and support Tangent mode. See G-Code Support .
M3 / M4 / M5	Additional spindles can be controlled using the parameter suffix M3.1 or M3.2 and likewise for the spindle reverse and stop command.