

Creality K2 Plus X CFS Creality K2 Plus X CFS [SHOP NOW](#)[Search](#)[SIGN IN](#)[CART\(0\)](#)

RON Lei ▾

Grbl V1.1 Quick Reference

[Shop](#) / [News](#) / Grbl V1.1 Quick Reference

Posted by SainSmart on January 5, 2022

Grbl V2.0 Quick Reference

If you cannot remember if \$110 or \$120 is the maximum rate or acceleration setting

[READ NOW →](#)



Provided by sainsmart customer

This is designed as an Aide Memoir for when I cannot remember if \$110 or \$120 is the maximum rate or acceleration setting, what error 31 means..... If you are changing the Grbl settings please refer to other sources as well.

[Grbl Error Codes](#)

[Grbl Alarm Codes](#)

[Grbl Non Gcode Commands](#)

[Grbl Settings](#)

[Grbl Supported GCodes](#)

[Grbl Gcode Parameters](#)

[Grbl States](#)

Grbl Error Codes

Code	Description
1	GCode Command letter was not found.
2	GCode Command value invalid or missing.

RECENT ARTICLES



Using the 4040 RENO CNC For Inventics Projects

November 13, 2024



CNC and Laser Software Compatibility Guide

November 5, 2024



Review of the SainSmart Genmitsu 3030-PROVer MAX

October 31, 2024



Spooky Season CNC Projects: Using the 4040 PRO MAX to Create Halloween Decorations

October 30, 2024



710W CNC Spindle Motor

3	Grbl '\$' not recognized or supported.
4	Negative value for an expected positive value.
5	Homing fail. Homing not enabled in settings.
6	Min step pulse must be greater than 3usec.
7	EEPROM read failed. Default values used.
8	Grbl '\$' command Only valid when Idle.
9	GCode commands invalid in alarm or jog state.
10	Soft limits require homing to be enabled.
11	Max characters per line exceeded. Ignored.
12	Grbl '\$' setting exceeds the maximum step rate.
13	Safety door opened and door state initiated.
14	Build info or start-up line > EEPROM line length
15	Jog target exceeds machine travel, ignored.
16	Jog Cmd missing '=' or has prohibited GCode.
17	Laser mode requires PWM output.



Review

October 25, 2024



Fusion 360 Guide for Genmitsu CNCs

October 18, 2024

CATEGORIES

- 3D Printing
- bit of the week
- Buyer Guides
- CNC
- Game
- Guest Blog
- How-To
- Laser
- Product Review
- Projects
- SAINSMART x Makers
- TPU
- Tutorials
- WIP

20	Unsupported or invalid GCode command.
21	> 1 GCode command in a modal group in block.
22	Feed rate has not yet been set or is undefined.
23	GCode command requires an integer value.
24	> 1 GCode command using axis words found.
25	Repeated GCode word found in block.
26	No axis words found in command block.
27	Line number value is invalid.
28	GCode Cmd missing a required value word.
29	G59.x WCS are not supported.
30	G53 only valid with G0 and G1 motion modes.
31	Unneeded Axis words found in block.
32	G2/G3 arcs need ≥ 1 in-plane axis word.
33	Motion command target is invalid.
34	Arc radius value is invalid.
35	G2/G3 arcs need ≥ 1 in-plane offset word.

36	Unused value words found in block.
37	G43.1 offset not assigned to tool length axis.
38	Tool number greater than max value.

Grbl Alarm Codes

Code	Description
1	Hard limit triggered. Position Lost.
2	Soft limit alarm, position kept. Unlock is Safe.
3	Reset while in motion. Position lost.
4	Probe fail. Probe not in expected initial state.
5	Probe fail. Probe did not contact the work.
6	Homing fail. The active homing cycle was reset.
7	Homing fail. Door opened during homing cycle.
8	Homing fail. Pull off failed to clear limit switch.
9	Homing fail. Could not find limit switch.

Grbl Non Gcode Commands

Code	Description
\$\$	Display Grbl Settings.
\$x=val	Change Grbl Setting x to val.
\$#	View GCode Parameters.
\$G	View GCode parser state.
\$C	Toggle Check Gcode Mode
\$H	Run Homing Cycle
\$J=rcode	Run Jogging Motion.
\$X	Kill Alarm Lock state.
\$I	View Build Info
\$N	View saved start up code
\$Nx=line	Save Start-up GCode line (x=0 or 1) There are executed on a reset.
\$RST=\$	Restores the Grbl settings to defaults.
\$RST=#	Erases G54-G59 WCS offsets and G28/30 positions stored in EEPROM.
\$RST=*	Clear and Load all data from EEPROM.
\$SLP	Enable Sleep mode.
Ctrl-x	Soft Reset
~	~ . . .

?	Status report query.
~	Cycle Start/Resume from Feed Hold, Door or Program pause.
!	Feed Hold – Stop all motion.

NOTE: There are a number of other real time commands using non typeable characters. These are for developers and are not normally used outside from the console. These control things such as feed overrides. And are not covered here.

Grbl Settings

Setting	Description
\$0	Step pulse, microseconds
\$1	Step idle delay, milliseconds
\$2	Step port invert, XYZmask*
\$3	Direction port invert, XYZmask*
<i>The direction each axis moves.</i>	
\$4	Step enable invert, (0=Disable, 1=Invert)
\$5	Limit pins invert, (0=N-Open. 1=N-Close)
\$6	Probe pin invert, (0=N-Open. 1=N-Close)
<small>status report '2' status 0-WCS position</small>	

\$10	Status report, : status. 0-WCS position, 1=Machine position, 2= plan/buffer and WCS position, 3=plan/buffer and Machine position.
\$11	Junction deviation, mm
\$12	Arc tolerance, mm
\$13	Report in inches, (0=mm. 1=Inches)**
\$20	Soft limits, (0=Disable. 1=Enable, <i>Homing must be enabled</i>)
\$21	Hard limits, (0=Disable. 1=Enable)
\$22	Homing cycle, (0=Disable. 1=Enable)
\$23	Homing direction invert, XYZmask* Sets <i>which corner it homes to.</i>
\$24	Homing feed, mm/min
\$25	Homing seek, mm/min
\$26	Homing debounce, milliseconds
\$27	Homing pull-off, mm
\$30	Max spindle speed, RPM
\$31	Min spindle speed, RPM
\$32	Laser mode, (0=Off, 1=On)
\$100	Number of X steps to move 1mm

\$101	Number of Y steps to move 1mm
\$102	Number of Z steps to move 1mm
\$110	X Max rate, mm/min
\$111	Y Max rate, mm/min
\$112	Z Max rate, mm/min
\$120	X Acceleration, mm/sec ²
\$121	Y Acceleration, mm/sec ²
\$122	Z Acceleration, mm/sec ²
\$130	X Max travel, mm <i>Only for Homing and Soft Limits.</i>
\$131	Y Max travel, mm <i>Only for Homing and Soft Limits.</i>
\$132	Z Max travel, mm <i>Only for Homing and Soft Limits.</i>

* XYZmask is a value setting for the X Y and Z axes.

Change if an axis is moving in the wrong direction. Value will be 0-7. ** Reporting units are independent of the units set in the Gcode!

Grbl Supported GCodes

Code	Description
F	Set Feed rate in Units/min (See G20/G21).

G0	A Rapid positioning move at the Rapid Feed Rate. In Laser mode Laser will be turned off.
G1	A Cutting move in a straight line. At the Current F rate.
G2	Cut a Clockwise arc.
G3	Cut an Anti-Clockwise arc.
G4	Pause command execution for the time in Pnnn. <i>P specifies the time in seconds. Other systems use milliseconds as the pause time, if used unchanged this can result in VERY long pauses.</i>
G10L2	Sets the offset for a saved origin using absolute machine coordinates.
G10L20	As G10 L2 but the XYZ parameters are offsets from the current position.
G17	Draw Arcs in the XY plane, default.
G18	Draw Arcs in the ZX plane.
G19	Draw Arcs in the YZ plane.
G20	All distances and positions are in Inches
G21	All distances and positions are in mm
G28	Go to safe position. NOTE: If you have not run a homing cycle and have set the safe position this is very 'unsafe' to use

to use.	
G28.1	Set Safe position using absolute machine coordinates.
G30	Go to the saved G30 position.
G30.1	Set Predefined position using absolute machine coordinates, a rapid G0 move to that position will be performed before the coordinates are saved.
G38.2	Probe towards the stock, error on a failure.
G38.3	As G38.2, no error on failure
G38.4	As G38.2 but move away, stop on a loss of contact.
G38.5	As G38.4, no error on failure.
G40	Cutter Compensation off. <i>Grbl does not support cutter compensation.</i>
G43.1	Dynamic Tool length offset, offsets Z end of tool position for subsequent moves.
G49	Cancel Tool length Offset.
G53	Use machine coordinates in this command.
G54	Activate the relevant saved origin.
G55-59	As G54, activates a different saved position

G61	Exact Path mode. <i>Grbl does not support any other modes.</i>
G80	Canned Cycle Cancel. <i>Grbl does not support any of the canned cycle modes which this cancels so it does nothing.</i>
G90	All distances and positions are Absolute values from the current origin.
G91	All distances and positions are Relative values from the current position.
G91.1	Sets Arc incremental position mode
G92	Sets the current coordinate point, used to set an origin point of zero, commonly known as the home position.
G92.1	Reset any G92 offsets in effect to zero and zero any saved values
G93	Inverse time motion mode.
G94	Units/min mode at the current F rate.
M0	Pause.
M1	As M0 but only pauses if an optional stop switch is on.
M2	Program End, turn off spindle/laser and stops the machine.
M3	Start spindle clockwise. In Laser mode sets Constant power.

M4	As M3, In Laser Mode sets Dynamic power.
M5	Stop the Spindle
M8	Coolant on as a flood. (Same as M7)
M9	Coolant off.
M30	Same as M2.
S	Set Spindle speed in RPM or Laser Power.

NOTE: Codes can contain leading zeros, G0 and G00 are the same. There are loads more GCodes, these are the ones Grbl supports. A lot of commands are Modal meaning they are remembered and applied to subsequent commands. For example, G0 X1 followed by Z5 remembers the G0 Mode and applies it to the Z5. S is modal, remembered from the last command. Two commands in the same modal group cannot be on the same line.

Grbl Gcode Parameters

Letter	Description
X Y Z	Distances or positions on the X Y Z axes.
I J K	Distances or positions for G2 and G3 Arcs. Correspond to the Z Y X axis respectively. These are always incremental coordinates regardless of G90/G91
L	Loop Cycle Count, supported but not used.

N	Line Number, supported but not used.
R	Arc radius for G2 and G3.
P	Multi-purpose parameter depends on command it is used in.
T	Tool selection, not used.

NOTE: All parameters must be followed by a number.

Grbl States

State	Description
Alarm	Homing enabled but homing cycle not run or error has been detected such as limit switch activated. Home or unlock to resume.
Idle	Waiting for any command.
Jog	Performing jog motion, no new commands until complete, except Jog commands.
Homing	Performing a homing cycle, won't accept new commands until complete.
Check	Check mode is enabled; all commands accepted but will only be parsed, not executed.
Cycle	Running GCode commands, all commands accepted, will go to Idle

	when commands are complete.
Hold	Pause is in operation, resume to continue.
Safety Door	The safety door switch has been activated, similar to a Hold but will resume on closing the door. <i>You probably don't have a safety door on your machine!</i>
Sleep	Sleep command has been received and executed, sometimes used at the end of a job. Reset or power cycle to continue.

For more information and details please refer to my 'Introduction to CNC for a Total Novice' series in the files section of the Facebook group '[SainSmart Genmitsu CNC Users Group](#)' or the Official documentation at <https://github.com/grbl/grbl>.

Grbl V2.0 Quick Reference

If you cannot remember if \$110 or \$120 is the maximum rate or acceleration setting

[READ NOW →](#)



Tags: [How-To](#)

« [Older Post](#)

[Newer Post](#) »

In Same Category

[Using the 4040 RENO CNC For Inventics Projects](#)

[CNC and Laser](#)

Related by Tags

[CNC and Laser Software Compatibility Guide](#)

[Uninterrupted](#)

[Software Compatibility Guide](#)

[Review of the SainSmart Genmitsu 3030-PROVer MAX](#)

[Spooky Season CNC Projects: Using the 4040 PRO MAX to Create Halloween Decorations](#)

[710W CNC Spindle Motor Review](#)

[Olympics Viewing with the Genmitsu APP for your CNC projects](#)

[3D Printed Mounts and Cable Clips](#)

[Genmitsu 4040-Reno, First Belt Driven CNC Router \(Assembly Guide\)](#)

[Grbl V2.0 Quick Reference](#)

[Crafting Custom Wooden Picture Frames with Your CNC Machine](#)

[PROVerXL 4030 V2 CNC, A Compact Powerhouse](#)

[Bit of the Week | Introducing 70V20 V-shape end mill bit set](#)

[How to use ER-11 collets of CNC Routers](#)

 Share

[Back to blog](#)

6 comments

I want to use the M06 or M25 command. How can this be implemented?

Yevgeniy •

AUGUST 15, 2024

i have grbl controlled machine and i want to use saftey door feature on it so how would is enable this feature kindly reply to me

akash pujari •

AUGUST 15, 2024

Hi!

GRBL makes me big fun. Only one problem – what about a parameter R with negative numbers? Thanks

Milan •

AUGUST 15, 2024

how do you dispay or save to file the current x,y and z co-ordinates.

Martyn Griffin •

SEPTEMBER 12, 2023

How Are You

Hello! Do you know if they make any plugins to protect against hackers? I'm kinda paranoid about losing everything I've worked hard on. Any suggestions?

<https://cutt.ly/c0xxgqa>

Best Regards

Kian Bindon •

SEPTEMBER 12, 2023

1 2

Leave a comment

Name *

Email *

Comment *

Please note, comments need to be approved before they are published.

Post comment

BE A POWER MAKER

Sign up for our newsletter now

Enter your email address

GO

SUPPORT

[Shipping](#)[Payment](#)[Return](#)[Warranty](#)[Track My Order](#)[Resource Center](#)[FAQ](#)[Accessibility](#)

ABOUT

[Our Story](#)[Privacy Policy](#)[Contact Us](#)[Genmitsu](#)[Genmitsu Ecosystem](#)[Blogs](#)[Share Your Work](#)[Sitemap](#)

SALE

[Special Offers](#)[Rewards Program](#)[Affiliate Program](#)[Referral Program](#)[Power Maker Recruit](#)[Reseller](#)[Education Discount](#)[Open Box & Clearance](#)

Trustpilot

© 2011 - 2024 Vastmind LLC. All Rights Reserved.