

Creality K2 Plus X CFS Creality K2 Plus X CFS

SHOP NOW



Sain SMART
POWER TO THE MAKERS



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 →](#)

grbl

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=gcode | 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 |
| ~ | Hard 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) |
| | Status report '2' status. 0=WCS position |

| | |
|-------|---|
| \$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 which corner it homes to. |
| \$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.</i> Other systems use milliseconds as the pause time, if used unchanged this can result in VERY long pauses. |
| 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 display 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

Leave a comment

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

Post comment

BE A POWER MAKER

Sign up for our newsletter now

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.