

CNC Singular

Operating Instructions - Original

CNC Singular	1
Operating Instructions - Original	1
Introduction	3
1.1 Purpose of the manual	3
1.2 Symbols used in the manual	3
1.3 Important notes:	3
Purpose and description of the machine	4
2.1 Purpose of the device	4
2.2 Basic controls:	4
2.3 Coordinate system	4
Health and safety	5
3.1 Work safety	5
Installation and commissioning	5
4.1 Device installation	5
Software installation:	5
5.1 USB driver installation	5
5.2 Candle software installation for Windows	5
5.3 Candle software settings	6
Basic principles of G-code	7
6.1 What it is a G-code	7
6.3 A-axis	8
6.4 B-axis	8
6.5 Listing of supported G codes by CNC Airbrush:	9
6.6 Important commands '\$'	9
How to write G-code	10
7.1 Simple G-codes	10
Example 1 - After switching on:	10
Example 2 - Nozzle cleaning:	11
Example 3 - Nozzle calibration	11
Software for creating more complex G-codes:	12
8.1 - More complex gcode	12
External control unit:	13
Creating a control code external control unit:	13





15.1 creating a G-code:	13
Spare parts	14
Warranty a	14
List of all control codes of the CNC airbrush system:	15
Special codes for pump control	16
Error states and their solutions:	16
2.2 Next include...	17

1. Introduction

1.1 Purpose of the manual

This manual is intended to provide the necessary information about the device to all persons who may come into contact with it during its operation, operation or maintenance. Each of these persons must be thoroughly acquainted with its contents before beginning any activity with the equipment.

1.2 Symbols used in the manual

Symbol	Meaning
	These symbols mean "Warning" and "Warning". They draw attention to facts that may cause damage to the device and / or the health of the user.
	The symbol indicates an important instruction, feature, procedure, or matter that must be followed during operation and maintenance of the equipment.
	The symbol indicates useful information.
	The symbol is a reference to another chapter of this manual.

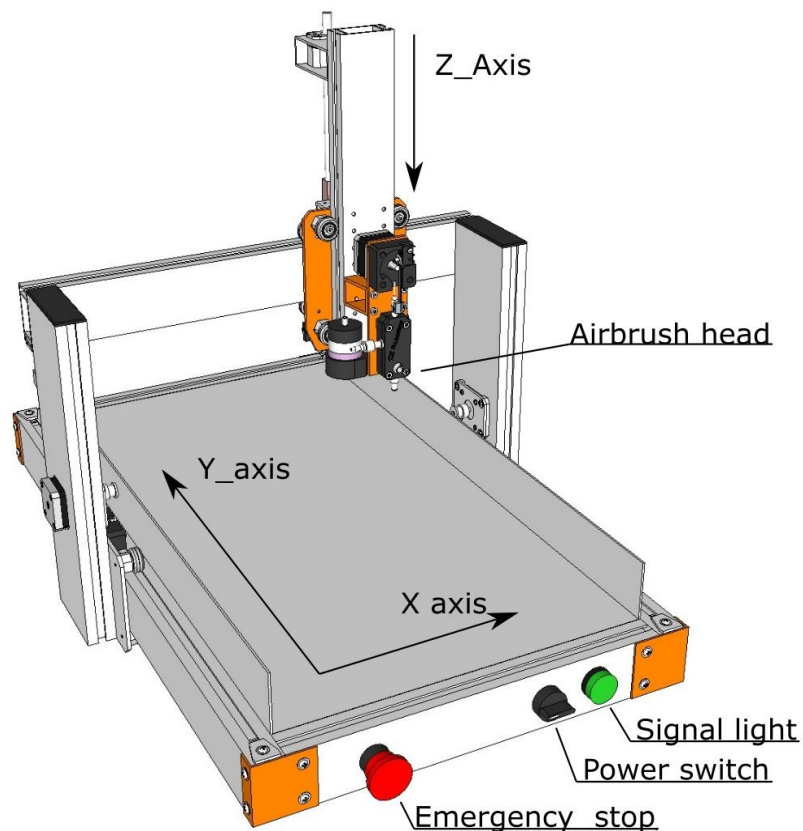
1.3 Important note:

Do not start up the machine until you have thoroughly read all the instructions.

2. Purpose and description of the machine

2.1 Purpose of the machine

CNC Singular is a CNC router designed for precise positioning of a working tool, typically an Airbrush spray head. The CNC Singular equipped with an Airbrush head is designed for precise application of liquids by means of air spraying. Thanks to the high positioning accuracy and smooth movement, it is possible to create very homogeneous surfaces, or to carry out selective application in virtually any shape



2.2 Basic controls:

The figure describes the three basic elements on the machine panel.

- Red emergency stop button
- Rotary switch on and off
- Signal light signaling that the machine is switched on

2.3 Coordinate system

The directions and orientations of the individual axes are also indicated in the figure. The Z axis has a starting zero point at the top and it is therefore necessary to enter the z coordinates with a negative sign.

3. Health and safety

3.1 Work safety

- All maintenance or cleaning work on the machine must only be carried out when the machine is switched off.
- Before switching on, make sure that the air inlets and the spray liquids are connected correctly and that nothing prevents the free movement of the portal.
- Never reach into the working area of the machine while it is in operation.
- The machine is powered by a safe voltage of 24V resp. 19V. use only a certified power supply.

4. Installation and commissioning

4.1 Installation of the device

Assembly should be carried out by two people.

You will need a Phillips 3 Phillips screwdriver.

- **Follow the video instructions here:** https://youtu.be/_M1YIZbaBLk
- Carefully remove the device from the transport packaging.
- Place the device on a level surface so that you have access from all sides of the device if possible during installation.
- Remove all polystyrene blocks

- Move the portal manually to the front of the instrument.
- Lift the rear of the instrument so high that you have access to the two connectors and the electrically operated valve at the bottom of the instrument.
- Connect the power, USB cable and air supply hose.

5. Software installation:

5.1

USB driver installation The first step is to install the usb driver for communication with the computer.


- Download the driver here. <http://work.czrobotics.cz/CH341SER.ZIP>
- Installation is performed in the usual way of running the SETUP.EXE or DRVSETUP64.EXE file

5.2 Installation of Candle software for Windows

- Now download and install the control software: https://github.com/Denvi/Candle/releases/download/v1.1/Candle_1.1.7.zip
- Install the Candle software in the usual way.

5.3 Candle software settings

- Start the software and select *Settings*
- Select port and set the speed to 115200 Baud

	<p>After switching on the machine resp. its start is always instate <i>ALARM</i>. Press thebutton <i>Unlock</i>. The software enters thestate <i>IDLE</i>. Only then is the machine ready to execute commands.</p>
---	--

Now we can try toin the command line in the lower right corner of the screen enter the first commands manually. The command is executed after pressing thebutton *Send*:

- \$ H command Performs the homing process automatically. Type \$ H and press *Send*. The machine first finds the initial position of the Z axis by moving up. Then it searches for the initial position of the nozzle control. If it is implemented in your machine, this is followed by a search for the starting position for air control or nozzle tilting. then the machine finds the starting position in the X-axis and finally in the Y-axis.
- After completing the homing process, it is always necessary to define the coordinates at which the nozzle is in the starting position. The G92 command is used for this. For example, for a Singular machine with a Smooth head in the basic configuration, use the G92X0Y520Z0A0 command. This ensures zero all the coordinates except Y coordinate, which is set a distance value from zero if the base configuration is 520 mm in the case of an extended version with the designation L or XL's 825 mm

In order not every time you turn manually enter all commands, we in the Candle software, use the User commands buttons. Settings for the User command Button are made in the menu Service -> Settings -> User commands. In the Button 1 line, enter the following command:

Machine type and configuration	Command
Singularwith Smooth Head (Basic Configuration)	\$ H; G92X0Y520Z0A0
Singular with head Double action	\$ H; G92X0Y520Z0A0B0
Singular with Tricolor head	\$ H; G92X0Y520Z0A0B0C0
Singular with Smooth head and Tilt system	\$ H; G92X0Y520Z0A0B-20
Singular L or XL with Smooth head	\$ H; G92X0Y825Z0A0
Singular L or XL with Double action head	\$ H; G92X0Y825Z0A0B0
Singular L or XL with Tricolor head	\$ H; G92X0Y825Z0A0B0C0
Singular L or XL with Smooth and Tilt head	\$ H; G92X0Y825Z0A0B-20

Individual commands that we would otherwise send to the machine separately can be separated from each other by the character "; "

In a Tilt configuration, it is possible to set a more accurate initial nozzle vertical position by changing the B value. To find the correct value for B:

- perform the homing process with the command \$ H
- set the default coordinates with the command G92X0Y520Z0A0B0
- to see and head better, move the portal to the front of the machine with the command G0 X150 Y50
- now enter the command G0 B20.
- check that the nozzle head points vertically downwards. If the head is tilted, we can correct its position with commands such as G0 B19.5 or G0 B20.2, etc.... Once we find the correct value for the vertical position of the head, we use it with a negative sign in the command to set the default coordinates. You must use a period as a decimal separator in all code statements.

You can now test machine movements from the command line.

- make sure the machine is turned on and pressin the Candle software *Button 1*. The machine starts the Homing process and finds the starting position in a slow motion.
- type G0 X100 at the bottom right of the command line and send to the machine. The machine travels at maximum speed to a position of 100mm to the right of the starting point.
- next command G1 X0 F1500 returns the nozzle in a slow motion (1500mm / minute, ie 25mm / s) to the default coordinate X0.

There are several other types of free downloadable software that can be used to control a GRBL machine. for example, the UGS Platform.

6. Basic principles of G-code

6.1 What is G-code



A detailed but general description of G-code is hereThere
<https://en.wikipedia.org/wiki/G-code>

are many variants of G-code, but one the line of code corresponds to one movement of the machine. The coordinates entered in the G0 or G1 command indicate the point in the XYZ area where the airbrush will move from its current position. The movements are performed sequentially, creating a sequence of movements. Thus, the G-code describes line by line the list of coordinates in the XYZ space where the airbrush nozzle will move. The complete structure of G-code programming is described in many other publications.

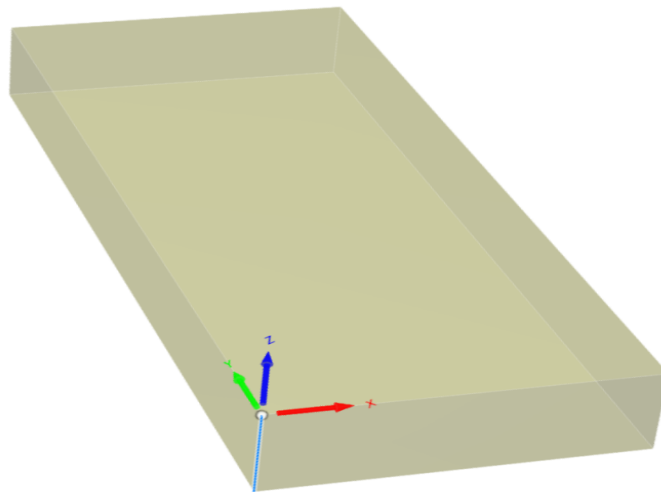
In this manual, we will describe a few basic commands that allow you to fully control the CNC Singular with Airbrush head.

6.2 Workspace - XYZ axes

We start from the basic XYZ coordinate system, where when looking at the machine from the front, the coordinate is X Movement in the machine width direction (left and right), Y coordinate movement in the machine length direction (forward and backward) and Z coordinate up and down.



Note: The initial position of the nozzle is at the top, ie when the nozzle is moved to the working position, it moves downwards and the coordinates of the Z axis must be entered with a minus sign.



6.3 A-axis

For easy control of the Airbrush nozzle, the system works with additional A-axis. While the values for the XYZ axes are given in millimeters, the values used for the A and B axes are dimensionless.

The values for the A axis range from 0.0 to 44.9 with a resolution of 0.1. They indicate the degree of nozzle opening. A value of zero means a closed nozzle, a value > 40 means a fully open nozzle. In practice, the opening of the nozzle occurs at a value of about 10 and the full opening of the nozzle occurs at values of about 30. This is due to the mechanical construction.

6.4 Axis B

The **Double Action system** uses the B axis to open the air valve inside the Airbrush gun. The value for the B axis ranges from 0-90. For design reasons, much of this range is fully open or fully closed. The air flow is regulated in the middle of this range. However, it is possible to set the air valve opening value with a resolution of 0.1 and thus control the air flow through the nozzle.

Machines with **the Tilt system** use the B axis to control the tilt of the nozzle resp. whole heads. Values range from approximately -20 to +20 with a resolution of 0.1. A value of 0 means that the nozzle is pointing vertically down. A value of B-20 means that the nozzle is pointing at the maximum angle to the right. A value of B20 means that the nozzle is tilted at the maximum angle to the left. See above for the default setting of B.

6.5 List of supported G codes by the CNC Airbrush system:

Codes for movement	G0, G1, G2, G3 , G38.2, G38.3, G38.4, G38.5, G80
Coordinate system selection	G54, G55, G56, G57, G58, G59
Working plane selection	G17, G18, G19
Absolute / relative mode	G90, G91
Unit selection	G20, G21
Zero point setting	G92
Tool length offset	G43.1, G49 (not used)
Program mode	M0, M1, M2, M30
Air control	M3 , M4, M5
Cooling control	M7, M8, M9 (not used)

6.6 Important commands '\$'

\$ X	Exits Alarm state and goes to Idle state
\$ H	Performs a homing sequence. All axes, including the nozzle control and its tilting, come into contact and stop. Note that this function does not set the coordinates to zero. The G92 command is used to set the default coordinates.

6.7 Description of the most important G codes

G0 moves the machine at maximum speed from the current point to the entered coordinates.

Example: **G0 X100 Y50 Z-20** - This command at maximum speed from the current point to the coordinates X = 100mm Y = 50mm Z = -20. It is typically used for crossings with the nozzle closed to a new position.

G1 moves the machine at the working speed from the current point to the entered coordinates.

Example: **G1 X100 Y50 Z-20 F1000** - This command Transfers a speed of 1000 mm / min from the current point to the coordinates as in the previous case. it is typically used for open nozzle movements during application.

G2 and **G3** move around the arc. G2 clockwise and G3 counterclockwise. This command draws a circle segment angle of up to about 180 degrees, the starting point to point xy circular segment centered at a point IJ

Example: **G0 X100 Y100** (Set the nozzle to its initial position to start the arc.

Order **G2 X200 I50 J50 Y200 F1800** Renders semicircle centered at +50 mm In the direction of the X axis ayas the end point at the coordinates x 200 y200 at a speed of 1800 mm / min -> 30 mm / sec

M3 - Starts the air supply Air pressure> 1.2 BarFor its correct operation of the solenoid

M5 - Stops the air supply

G21 - This statement says that all dimensions for the XYZ axes in millimeters

G92 - Assigns the actual position value of the axis XYZa or B C

7. Writing G code

7.1 simple G-codes

for easy movements of the machine just any text editor such as WindowsNotepad.

Example 1 - After switching on:

\$ H	Homing process - an automatic process where the machine finds its starting position.The machine searches for the starting points of each axis by moving slowly along each axis
G21	It defines that the values XY Z are in millimeters
G92 X0 Y520 Z0 A0	Set the default coordinate values. For long machines with L and XL markings, is Y value set to 825. Since the limit switch for the Y-axis is located at the rear of the machine, the nozzle is located 520 or 825mm from the initial zero point of the working area after the homing process is completed. In the case of a machine with a Double Action head, add the B0 parameter. to set the default air cam value. This sets the default value for the air cam. In the case of a machine with a Tilt system, add parameter B-20 because the nozzle tilt is in the leftmost position.

Example 2 - Nozzle cleaning:

For this example we will need a wide neck container or bowl.

Type G0 X270 Y500 manually at the bottom right of the Candle software. The airbrush moves to the specified coordinates. Now place a wide-mouthed container or waste ink bottle under the nozzle. The following program will clean the nozzle.

G0 Z0 (Move nozzle to Top position)
G0 X270 Y500 (Move to the position where a waste ink container is. Add parameter **B0** if you are using the **Tilt system**)
G0 Z-50 (Move the nozzle down - closer to the waste ink container)
M3 (Open air solenoid)
(G0 B90) (Use the G0 B90 command and if you are using the **Double Action** system)
G0 A40 (Open nozzle to max value)
G0 A0 (Close nozzle)
G0 A40
G0 A0
G0 A40
G0 A0
G0 A40
G0 A0
G0 A40
G0 A0
G0 A40
G0 A0
M5 (Close solenoid)
G0 Z0 (Move nozzle to Top position)

Example 3 - Nozzle calibration

For this example we will need a printed calibration pattern Calibartion.pdf

G0 A0 (Close the nozzle as a precaution)
G0 X35 Y50 (Move to start position. Add parameter **B0** if you are using the **Tilt system**)
G0 Z-90 (set Z position - you can edit)
M3 (Open air solenoid)
(G0 B90) (Use the G0 B90 command and if you are using the **Double Action** system)
G1 F3000 (Set speed of work movment - unit mm / min)
G1 X255 A5 (Move nozzel to X250 and opnen nozzle)
G0 A0 (Close nozzle)
G0 X35 Y80 (Quick move to start of next test line)
G0 A5
G1 X255 A10
G0 A0
G0 X35 Y110
G0 A10
G1 X255 A15
G0 A0 (Close nozzle)
M5 (Close Air solenoid)
G0 Z0 (Move nozzle up)
G0 X100 Y350 (Go to final position)

8. Software for creating more complex G-codes:

8.1 - More complex gcode

Download graphic editor inkscape: <https://inkscape.org/cs/release/inkscape-0.92.4/>
and a plugin for working with the CNC Airbrush system: http://work.czrobotics.cz/Airbrush_Plugin.zip

- First, install the Inkscape software in the standard way.
- Copy the two files stored in the Airbrush_Plugin.zip archive (airbrush.inx and airbrush.py) to the directory. "*C: \ Program Files \ Inkscape \ share \ extensions*"
- Now the software is ready to create your first G code.

Tip: Draw lines and curves (trajectories of the Airbrush nozzle). Warning: some objects, such as fonts, must be converted to Path before exporting (menu Path -> Object to Path)

When you have lines and curves drawn open *Extention* menu -> CNC Airbrush G-code generator

Describe:

Nozzle Axis name: A

Open Air command: B60 or - nothing -

Open Close command: B0 or - nothing -

Travel speed: 8000 - not used yet. Machine maximum speed is used for crossing.

Drawing Speed: 1200mm / min

Nozzle: - degree of nozzle opening -

Start delay - nothing -

Z - axis: value from 0 to -120 (nozzle distance from the surface 0 = Top> max. distnace -120 = bottom)

Final parking Code: - nothing or eg G0 X200 Y0 Z0 -

Directory: - output folder (eg C: \ Godes) -

File name: - filename of the generated g-cod -

Append to a File: if checked, then the created g-code appends to the end of an existing file (suitable for Tricolor head)

9. External control unit:

7.1 Description

The control unit and the whole system are designed so that the normal operator has no possibility to change the system settings or modify the primer / activator application programs.

Only a person trained and thoroughly familiar with how to make settings or know how to create new programs may change the system settings to create new programs or edit existing ones.

7.2 Setting the system parameters:

Open the control unit and there is a button on the yellow PCB at the bottom left. (There is a safe voltage of 24V or lower inside the entire control unit) After a short press, the control unit goes into setting mode. Close the control unit and the display will show the setting options. Set the required parameters and press the button on the yellow printed circuit board again to return the machine to normal operating mode.

7.3 SD card

(Open the control unit and there is an SD card slot on the inside of the display module. Remove the SD card and connect it to the computer.

10. Creating a control code external control unit:

15.1 creating a G-code:



Program files must be named in 8.3 This means that the name of the file must contain 8 characters or numbers before the dot and 3 digits after the decimal point. for example NAME-987.TXT

special importance are the file names prefixed with an underscore

_Start. If a file with this name is present on the SD card, it will be executed automatically when the machine is turned on or restarted. It usually contains commands for both basic and initialization, ie starting a sequence to find the zero point of the default coordinates setting. It can also contain a command to start the pump and so on.

1. The file named **** and digits are sequences of commands or speed dial programs using the buttons on the right side of the control panel. Typically, commands **_1** through **_6** correspond to buttons 1 through 6 on the front panel. Here it is advisable to set various normally recurring process type for cleaning the break / shutdown nozzle and so on....

Files starting with the “**_**” character cannot be used other than for these special purposes.