

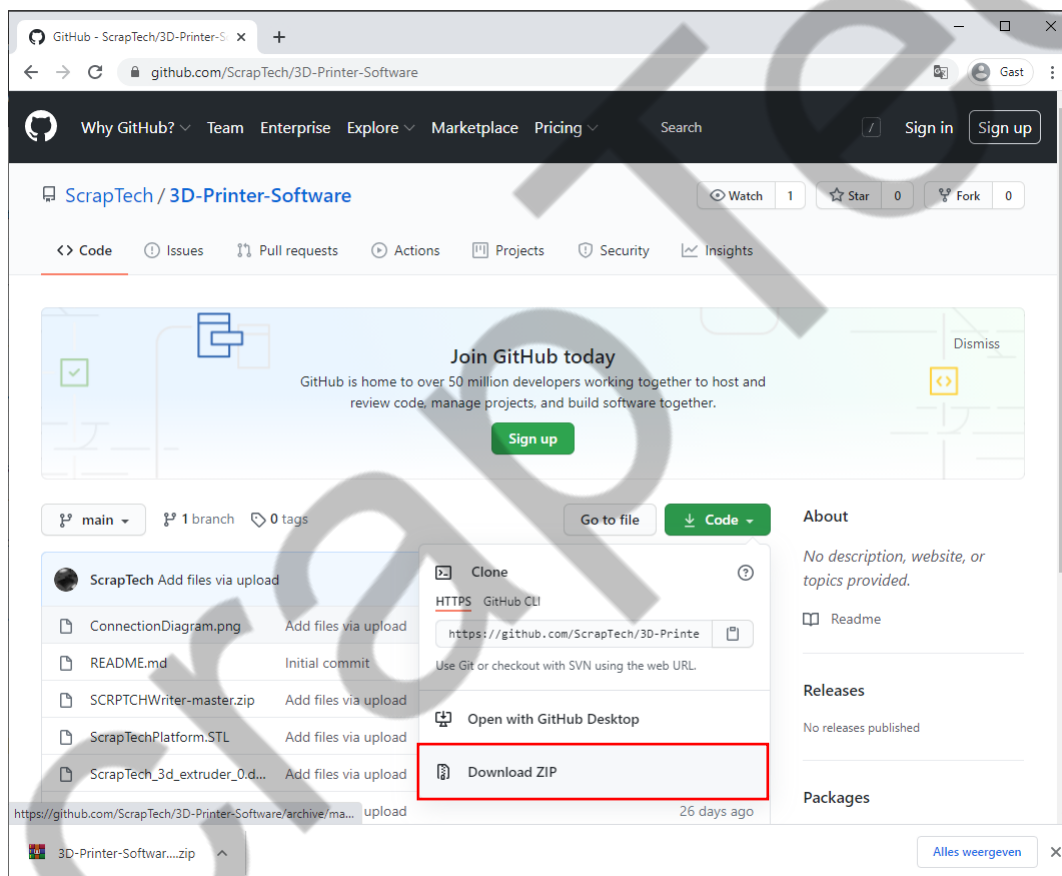
Installation of the 3D printer software

Installation of GRBL on Arduino

The first step is to prepare the Arduino to understand G-code. G-code is a programming language that gives instructions to CNC machines on how to create an object. G-code can be interpreted by GRBL, a high-performance motion control software that is lightweight enough to install on Arduino Uno.

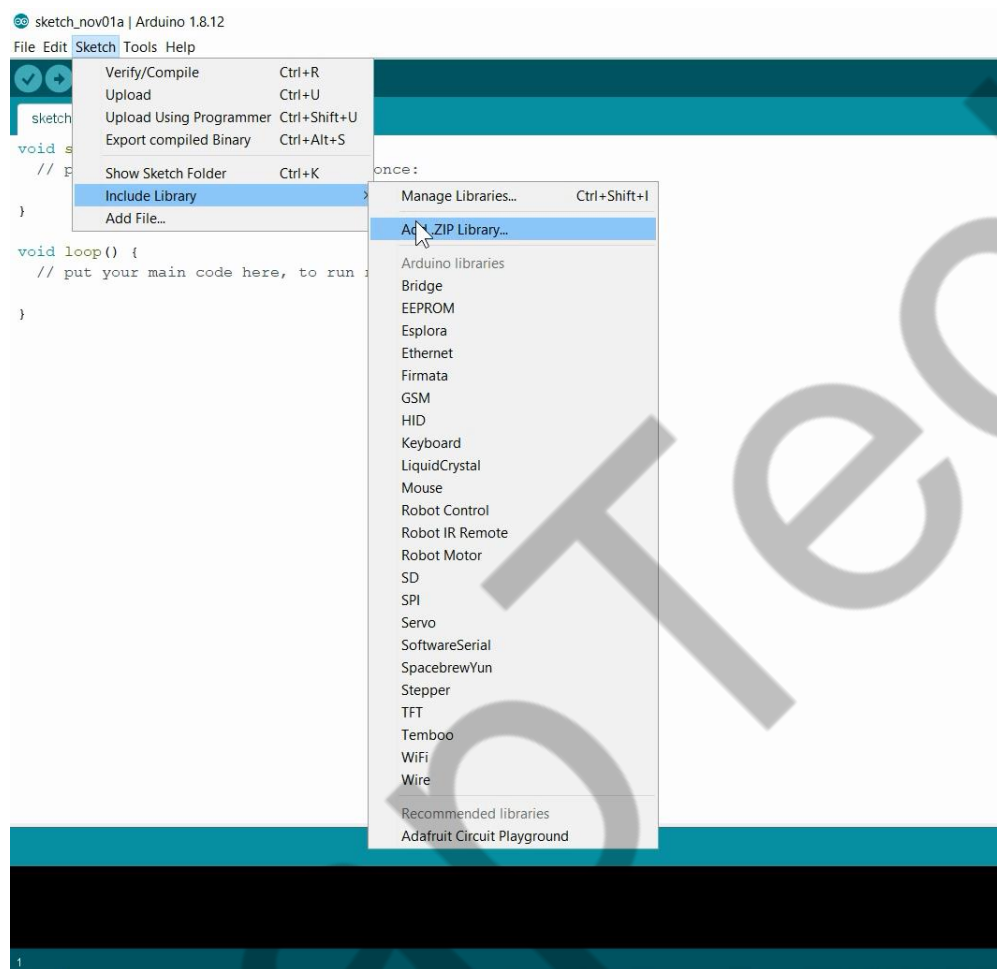
All software files used in this manual, including GRBL, can be downloaded via the link below.

<https://github.com/ScrapTech/3D-Printer-Software>



Press “Download ZIP” to download the complete folder. Once downloaded, extract the “.zip” to a known folder.

Start up the Arduino software to install GRBL on the Arduino. We will add GRBL as a library to Arduino.

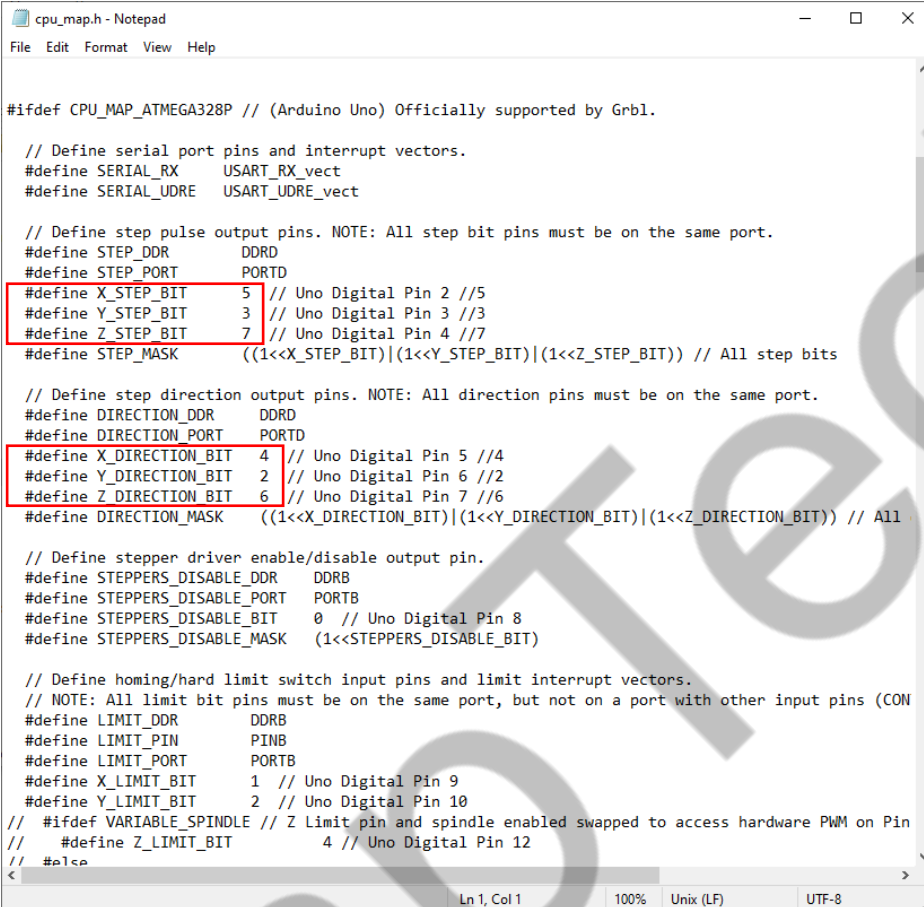


Browse to the location where you extracted the downloaded files and select “grbl.zip”. The library is now included in the software.

If you closely followed the steps from my previous videos, this library should work properly. When you made the connections differently, you will have to alter the pinout of the library.

The pinout of the library is defined in the file “cpu_map.h”, which you can find in the folder “C:\This PC\Documents\Arduino\libraries\grbl”.

You can open this file with Notepad, and if necessary, alter the pinout.



```
#ifndef CPU_MAP_ATMEGA328P // (Arduino Uno) Officially supported by Grbl.

// Define serial port pins and interrupt vectors.
#define SERIAL_RX    USART_RX_vect
#define SERIAL_UDRE  USART_UDRE_vect

// Define step pulse output pins. NOTE: All step bit pins must be on the same port.
#define STEP_DDR      DDRD
#define STEP_PORT     PORTD
#define X_STEP_BIT    5 // Uno Digital Pin 2 //5
#define Y_STEP_BIT    3 // Uno Digital Pin 3 //3
#define Z_STEP_BIT    7 // Uno Digital Pin 4 //7
#define STEP_MASK     ((1<<X_STEP_BIT)|(1<<Y_STEP_BIT)|(1<<Z_STEP_BIT)) // All step bits

// Define step direction output pins. NOTE: All direction pins must be on the same port.
#define DIRECTION_DDR  DDRD
#define DIRECTION_PORT PORTD
#define X_DIRECTION_BIT 4 // Uno Digital Pin 5 //4
#define Y_DIRECTION_BIT 2 // Uno Digital Pin 6 //2
#define Z_DIRECTION_BIT 6 // Uno Digital Pin 7 //6
#define DIRECTION_MASK ((1<<X_DIRECTION_BIT)|(1<<Y_DIRECTION_BIT)|(1<<Z_DIRECTION_BIT)) // All

// Define stepper driver enable/disable output pin.
#define STEPPERS_DISABLE_DDR  DDRB
#define STEPPERS_DISABLE_PORT PORTB
#define STEPPERS_DISABLE_BIT  0 // Uno Digital Pin 8
#define STEPPERS_DISABLE_MASK (1<<STEPPERS_DISABLE_BIT)

// Define homing/hard limit switch input pins and limit interrupt vectors.
// NOTE: All limit bit pins must be on the same port, but not on a port with other input pins (CON
#define LIMIT_DDR      DDRB
#define LIMIT_PIN      PINB
#define LIMIT_PORT     PORTB
#define X_LIMIT_BIT    1 // Uno Digital Pin 9
#define Y_LIMIT_BIT    2 // Uno Digital Pin 10
// #ifdef VARIABLE_SPINDLE // Z limit pin and spindle enabled swapped to access hardware PWM on Pin
// #define Z_LIMIT_BIT    4 // Uno Digital Pin 12
// #else
```

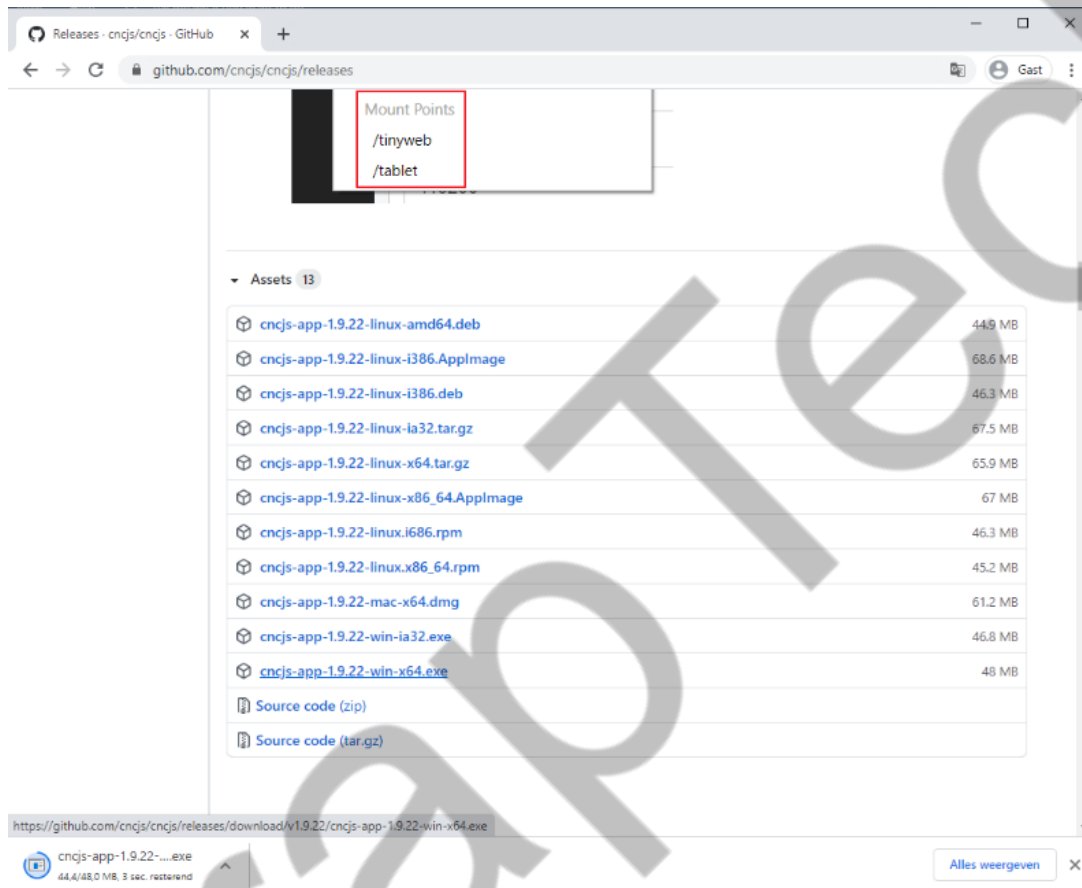
When the pinout is correct, we can upload the library into the Arduino. In order to do so, browse to the folder “C:\This PC\Documents\Arduino\libraries\grbl\examples\grblUpload”, open the file “grblUpload.ino” and click upload.

Installation of CNCjs on your computer

In the next step, we will install CNCjs, a program to send the G-code to the Arduino.

Simply download the latest version via the link below and install it.

<https://github.com/cncjs/cncjs/releases>

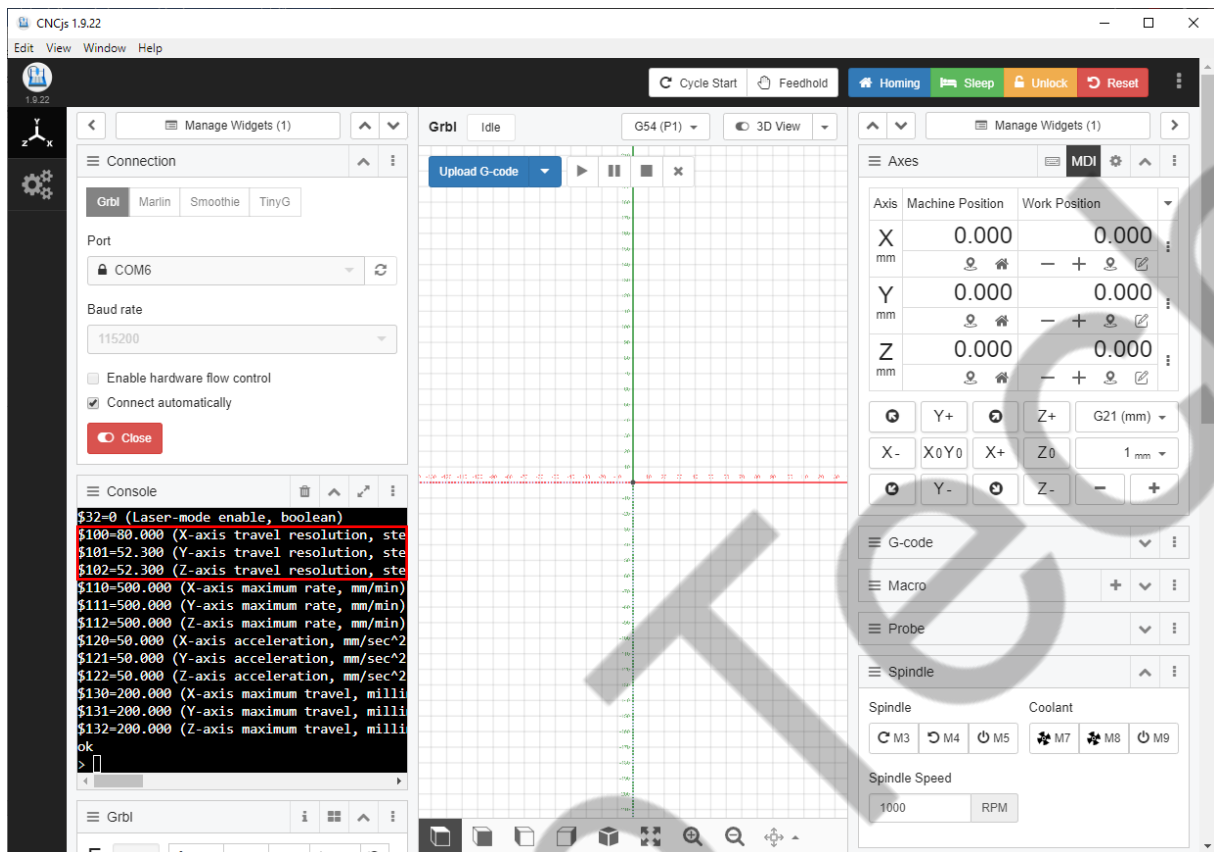


Once installed, you have to adjust the pitch of the axes of your printer. You can do this by changing the parameters in the Console section of CNCjs.

I used the following commands to adjust the parameters:

- \$100 = 80.0
- \$101 = 52.3
- \$102 = 52.3

with \$100, \$101 and \$102 being the commands and 80.0, 52.3 and 52.3 the pitch of respectively the X-, Y- and Z-axis.

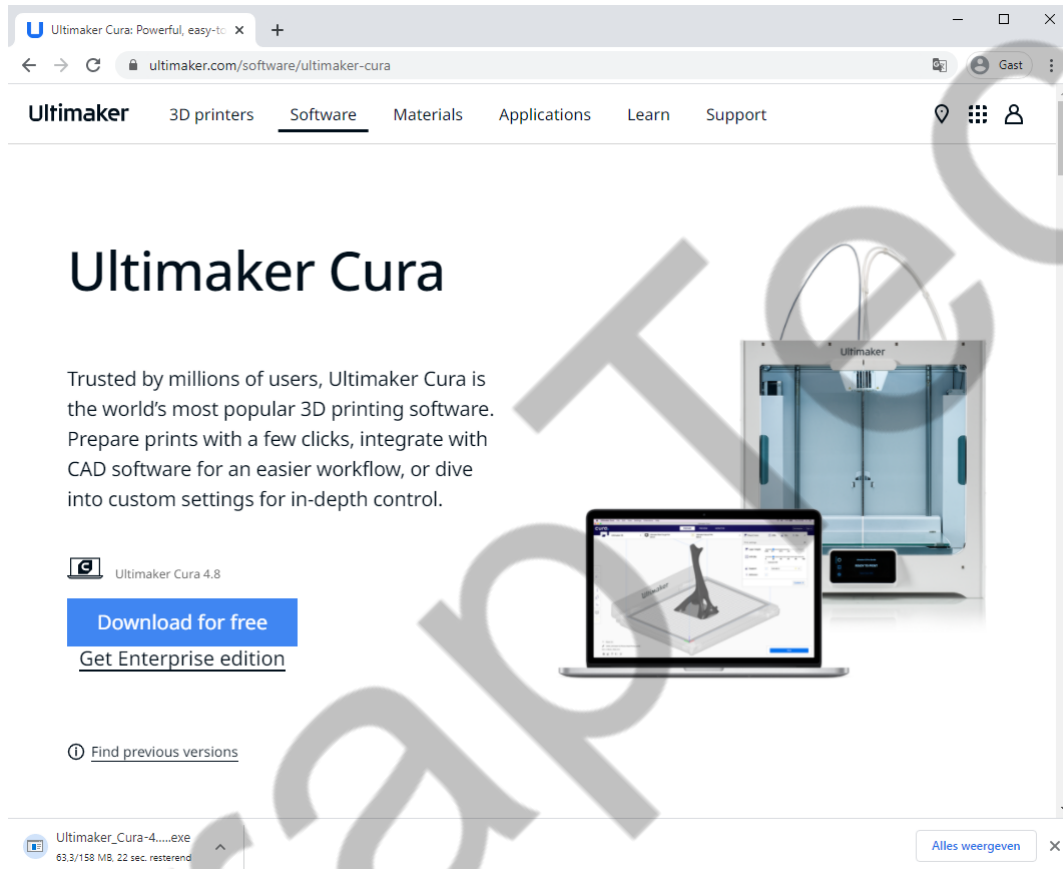


Installation of Cura on your computer

As final step, we install the slicing software, Cura. This software creates G-code from a 3D file. The software is developed by Ultimaker, who also makes excellent 3D printers.

The software can be downloaded via the link below.

<https://ultimaker.com/software/ultimaker-cura>



By default, Cura creates G-code that can be interpreted by Marlin and other motion control software, however, not by GRBL.

Therefore, we add some files in the installation folder in order to make Cura compatible with GRBL.

The folder downloaded and extracted in the first step, includes the following 4 files:

- ScrapTech_3d_extruder_0.def.json
- scraptech_3DPrinter.def.json
- ScrapTechPlatform.STL
- SCRPTCHWriter-master.zip

Go to the installation folder ("C:\Program Files\Ultimaker Cura 4.7") and paste the files in the correct subfolders according to the table below. Make sure you extract the file "SCRPTCHWriter-master.zip" first.

ScrapTech_3d_extruder_0.def.json	resources\extruders
scraptech_3DPrinter.def.json	resources\definitions
ScrapTechPlatform.STL	resources\meshes
SCRPTCHWriter-master (folder)	plugins

After completing this last step, you will be able to create the G-code and print any object as shown in this video:

<https://youtu.be/x13azJm-vqM>