Zonestar

How to install and use Laser engraving feature

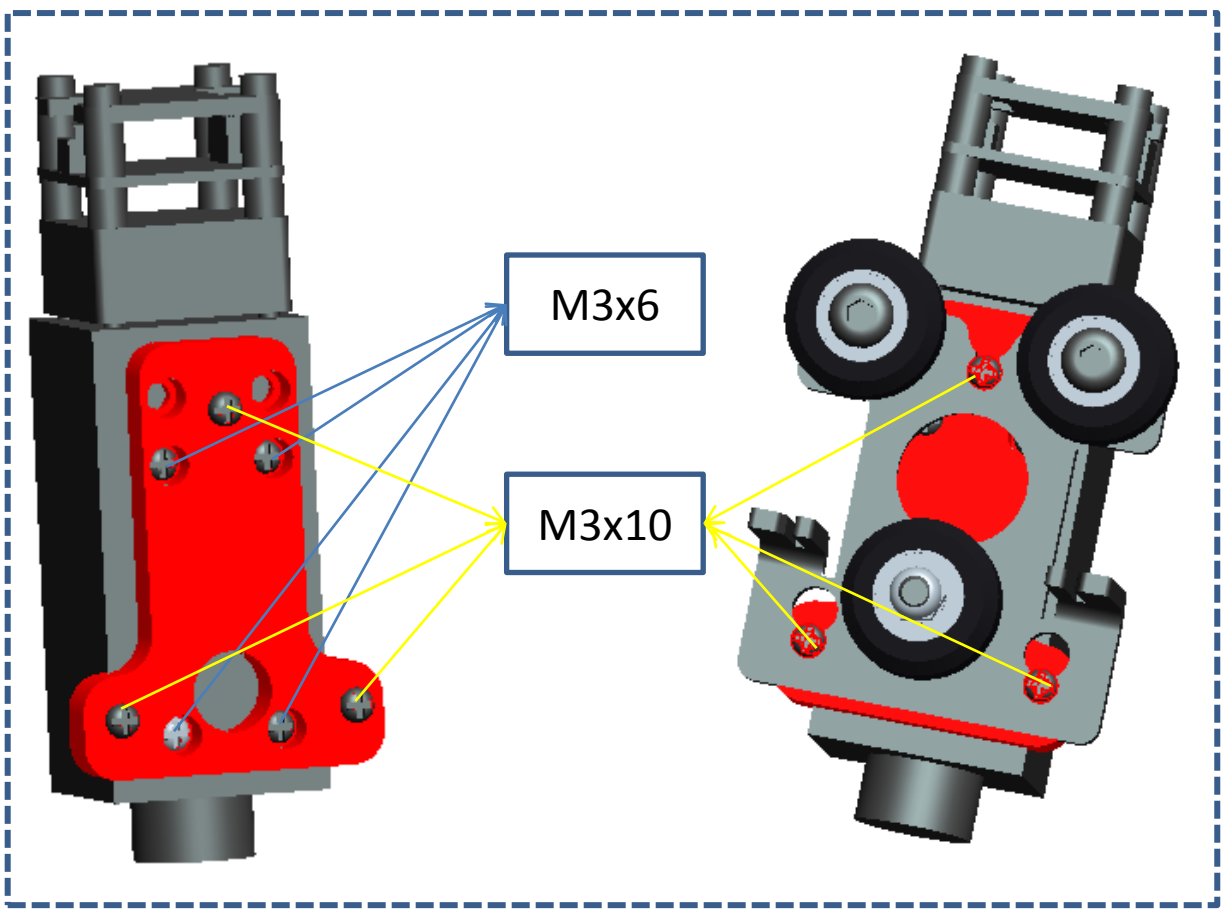
Ver 5.0

!!ATTENTION!!

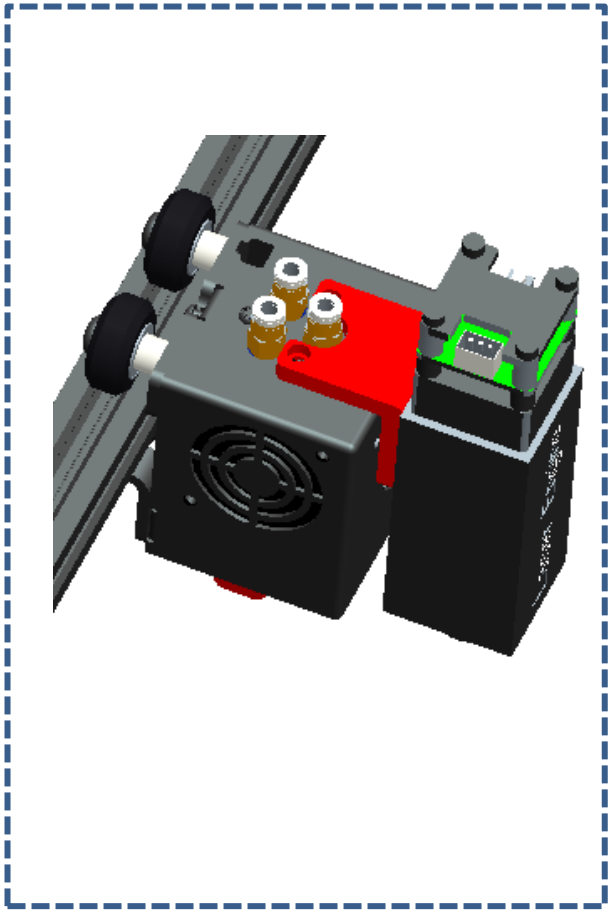
1. **Wear goggles before turning on the laser.**
2. **Do not place any part of the body within the range that the laser reached.**
3. **The laser may ignite the object being carved, PAY ATTENTION TO FIRE!**
4. **The work voltage of Laser cannot lower than the voltage of control board!**

Install the laser engine (Z8T/Z8P/Z9S/Z9F)

You need to print a bracket. The stl files of the print carriage is placed in the "Bracket stl file for install laser kit" directory.



Mounted on X carrire(recommended)



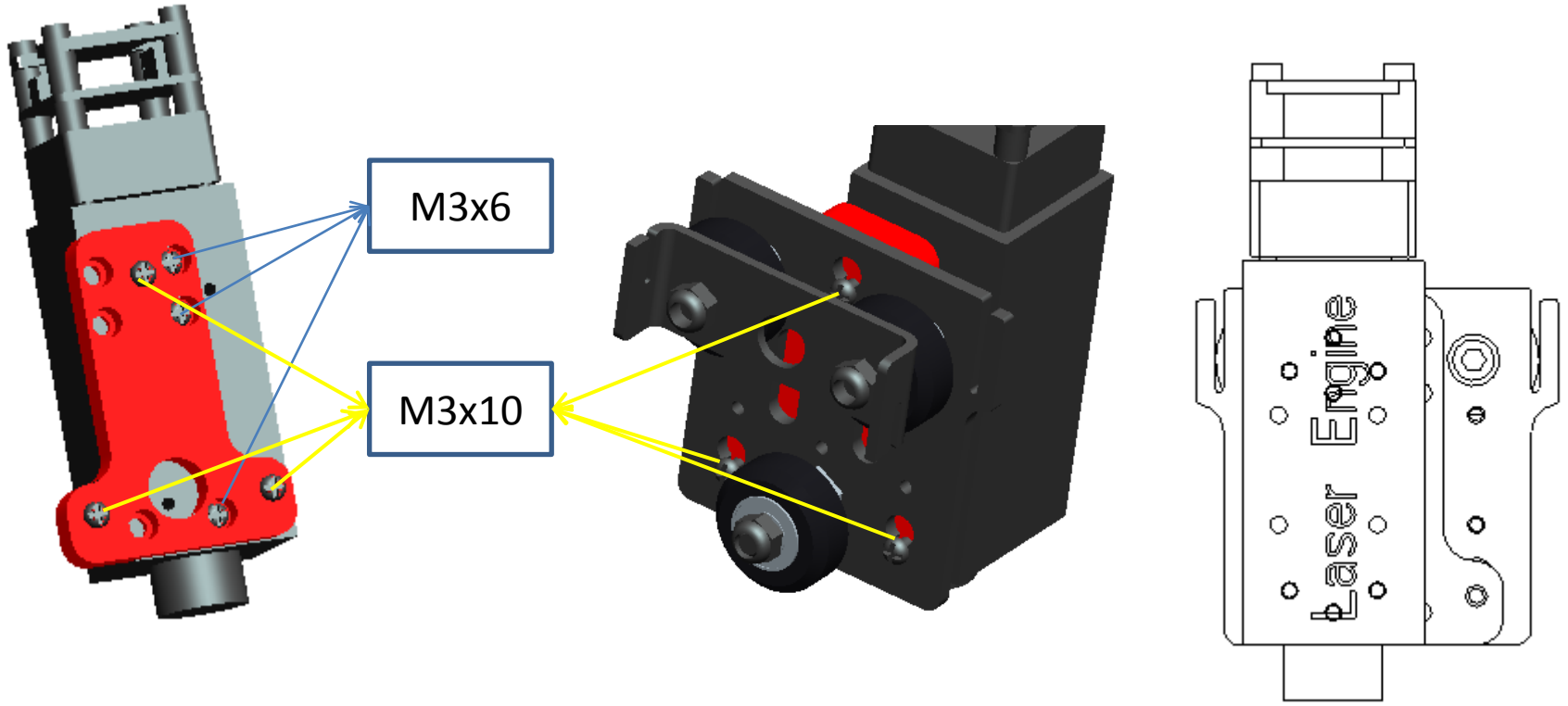
Mounted on X hotend

Zonestar

Install the laser engine (Z9V5)

You may need to print a bracket.
stl file name: LK25_LK55_BR_V5.stl
Please find it in here: "Bracket stl file for install laser kit\common"

LK25_LK55_BR_V5



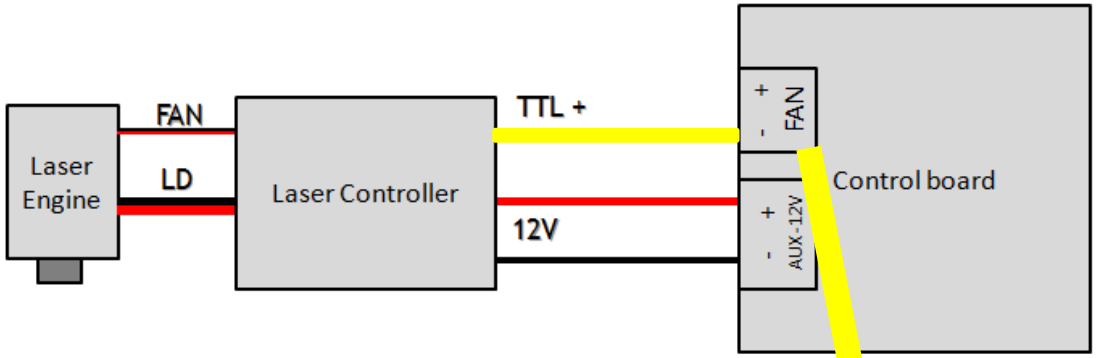
Zonestar

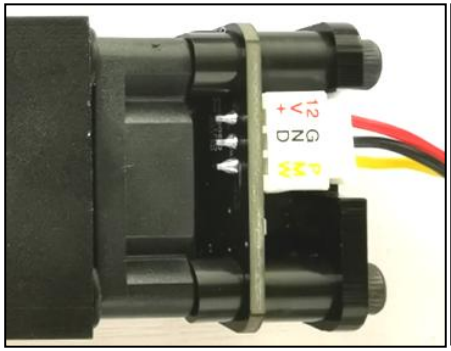
Wiring(Mode A)

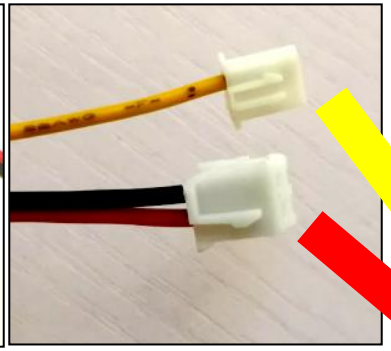
For ZMIB and the older version ZRIB (V3.x and V5.x) control board, refer to this wiring diagram.

Warning: Error to connect the power wire will burn the laser engine. Please pay attention to distinguish the +/- of the power wire.

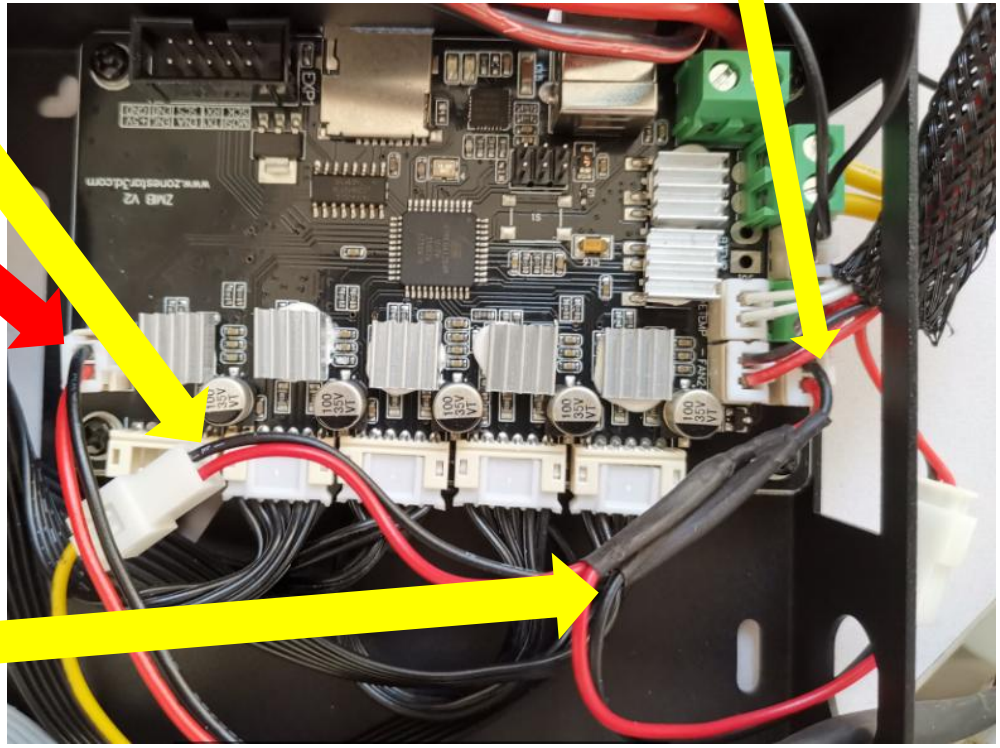
On Laser Engine		On Control Board
TTL	+	FAN -
	-	NC
12V	+	AUX_POW Connector or DC+
	-	AUX_POW Connector or DC-







On Laser Engine | **To Control Board**



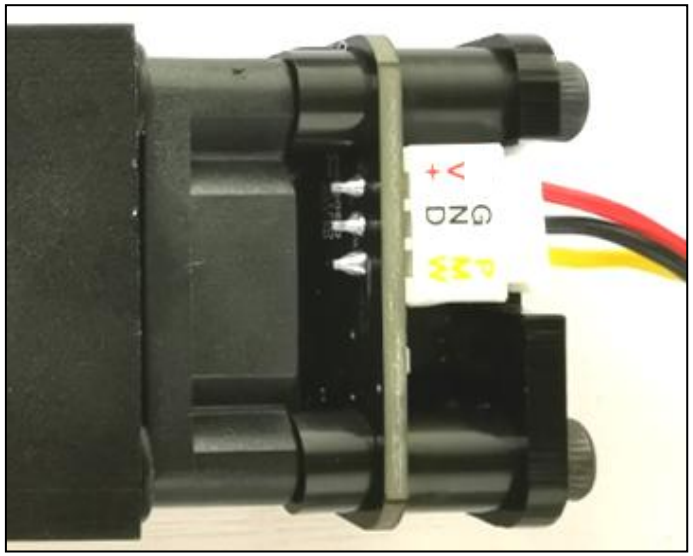
1. When using the print function, connect the extruder fan (on front of the printhead) to the control board's FAN1 socket. When using the laser engraving function, connect the laser-engraved TTL signal to the FAN1 socket.
2. If you need to use both laser engraving and printing, in the same time, you can connect the TTL signal of the laser head directly to the "-" line of the extruder fan.

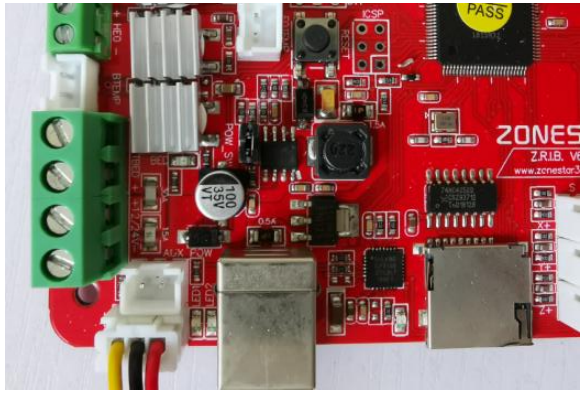
Zonestar

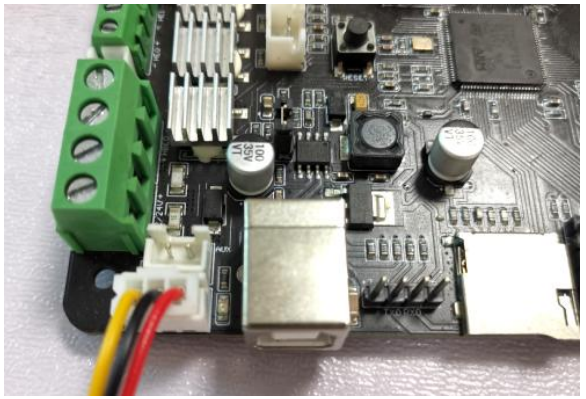
Wiring(Mode B)

There is a “Laser” connecor of Laser for ZRIBV6 control board, simply plug the wire to this connector.

On Laser Engine	On Control board
+	DC+
TTL	PWM
-	DC-







Zonestar

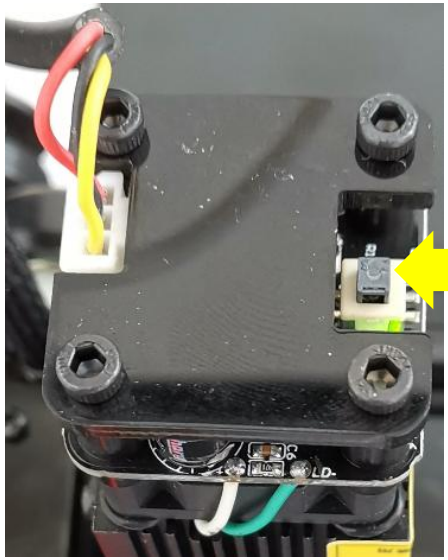
Adjust the focus

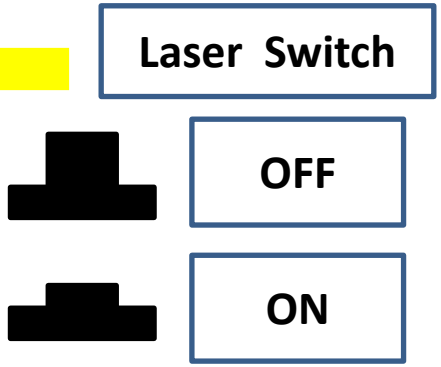
Please follow the steps below to adjust the focus:

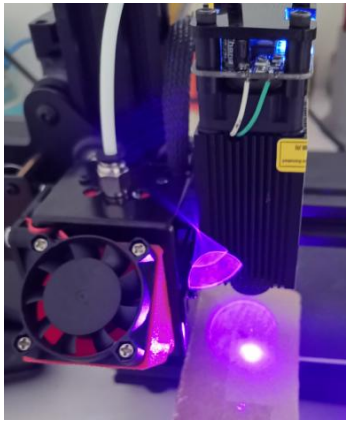
1. Turn off the laser (as shown below, the button is in the high position, a green light on the laser control board is on).
2. Operate the control panel of printer to HOME position (**MENU>>Prepare>>Auto HOME)**
3. Operate the control panel to move the nozzle to above about 10 mm on the hot bed, and the laser can be illuminated into the hot bed.
4. Place a piece of cardboard on the hot bed and adjust the fan speed to 0 **(Wiring Mode A)** or 255 **(Wiring Mode B)**.
5. Carefully rotate the lens at the front of the laser head to adjust the focus. Observe the center of the spot to minimize its diameter.
6. **Wear protective glasses** and turn on the laser output (refer to the below figure, the blue light is on).
7. Move the cardboard , pay attention to whether the engraving can be achieved, and if it is OK, turn off the laser output.

Note:

1. **Be careful when adjusting the focal length , the lens is easily damaged.**
2. **The focal length DOES NOT NEED to be very accurately, you can fine-tuned it by adjusting the Z axis height.**







Zonestar

Usage notes and tips

!!WARNING!!

Be careful when adjusting the focus, the lens is easier be damaged.

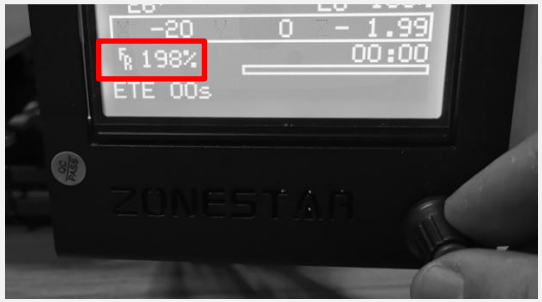
Turn the laser head power on / off.

When “Turn On Laser Power and Press Knob” is displayed on the LCD menu, turn on the laser engine and press the knob on the menu.

When the engraving is completed, turn off the laser engine.

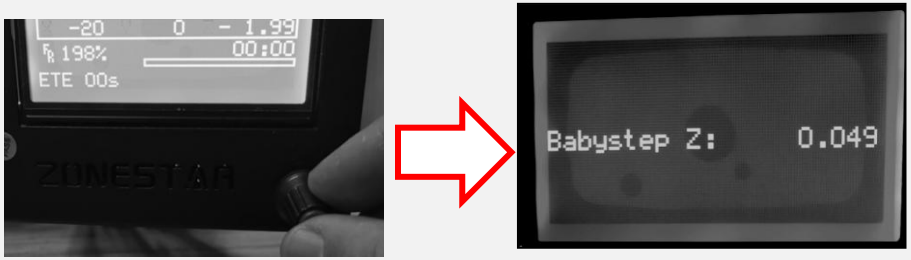
Adjusting speed during engraving.

Different speeds may be required when engraving (cutting) different materials, you can simply rotates the knob to adjust the feed rate(FR) to set the engraving speed (the response will be slightly delayed).



Adjusting focus during engraving.

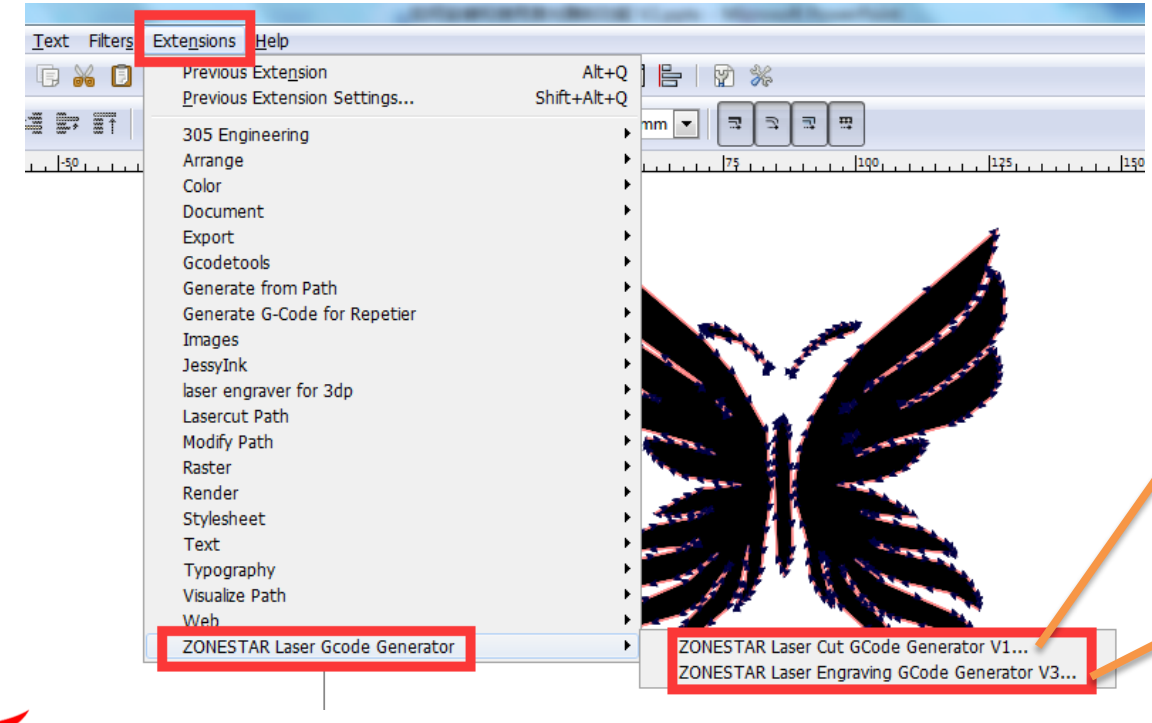
If the focal length is not very good during the engraving process, you can press the knob twice to open the Z-axis height fine-tuning menu to adjust the distance between the lens with objects.



Zonestar

Install software and plugins

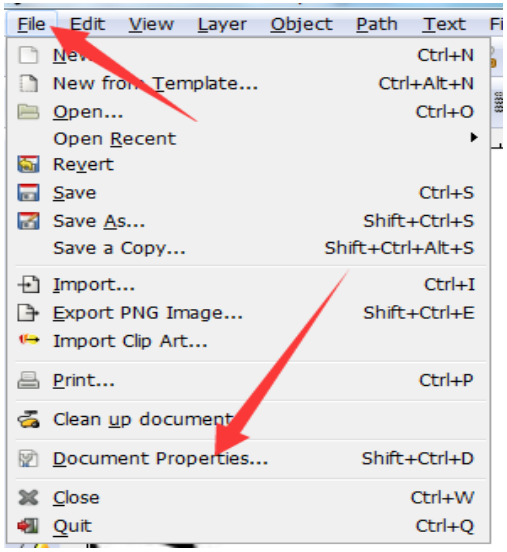
1. Download Inkscape, the download link is: https://inkscape.org/en/download/. recommended version is 0.92.0
2. Following the prompts to install Inkscape .
3. Install the plugin: Copy all the files under **Zonestar_Laser_Gcode_Gen** to the ***C:\Program Files\Inkscape\share\extensions*** (assuming your installation directory is ***C:\Program Files\Inkscape***)
4. Run Inkscape, in ***Extensions*** menu you can see 2 plugins. The one is ***ZONESTAR Laser Cut Gcode Generator,*** it can be used for laser cutting or to create simpler frame engraving patterns. Another is ***ZONESTAR Laser engraving Gcode Generator*** , it can be used to create engraved patterns with fills, as shown below.

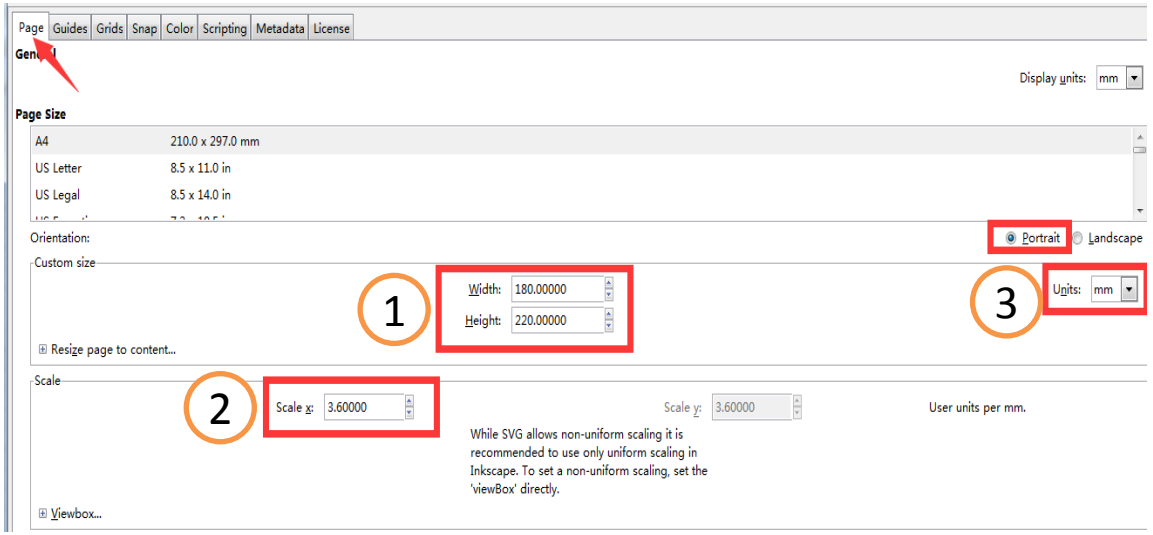


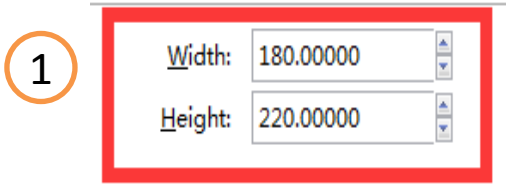
Zonestar

Generate gcode files using Laser Cut Gcode Generator

1. Setting

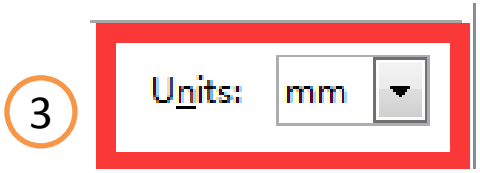






The width and height of the page are according to the actual engratable size, usually it is 40 to 50 mm smaller than the printer's building size in the X-axis and the length is the same as the Y axis. Anyway, it depends how to install the laser engine.

Set the scale to 3.6. If the actual size of the engraving is slightly different from the original size, you can modify this parameter to fine-tuning

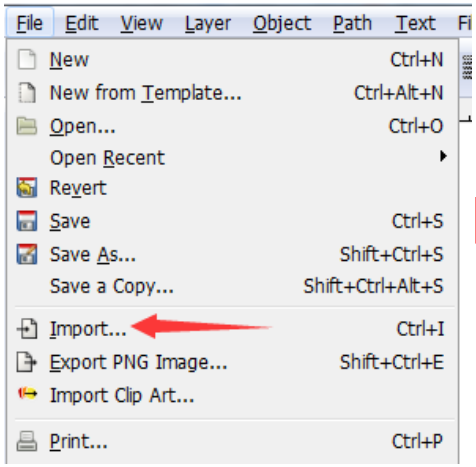


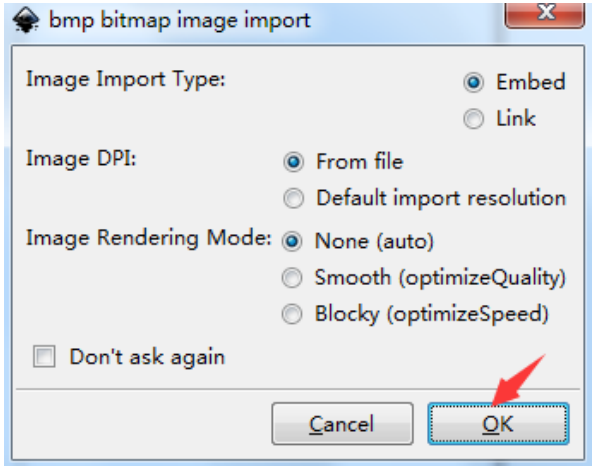
Set the size unit to mm.

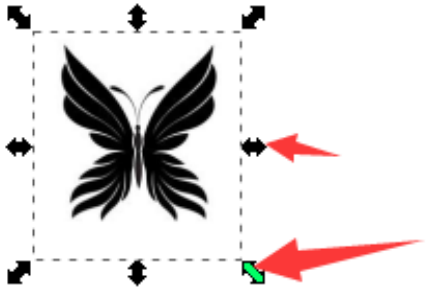
Zonestar

Generate gcode files using Laser Cut Gcode Generator

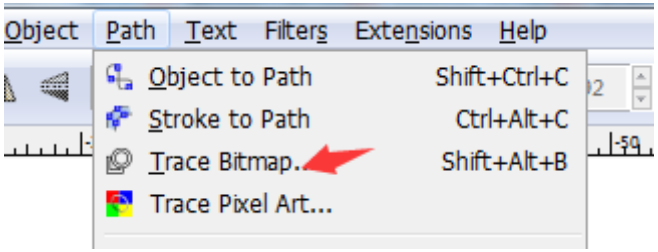
2. Import the image and zoom and place it in the position to be engraved.



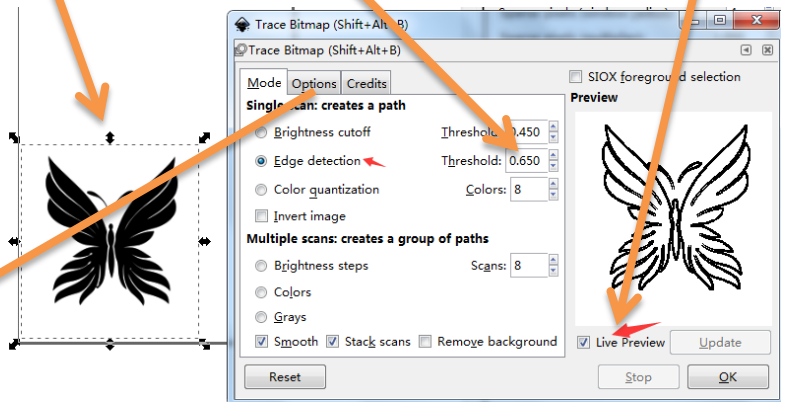


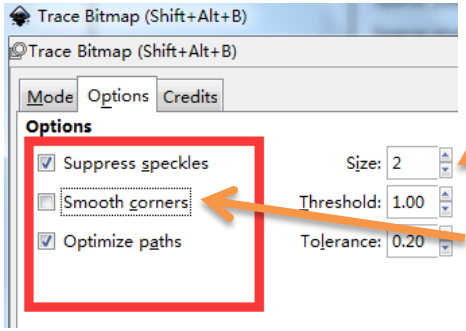


3. Generate the OD line



Click to select the picture	Adjust it to get a sharper border	Click Live Updata to preview



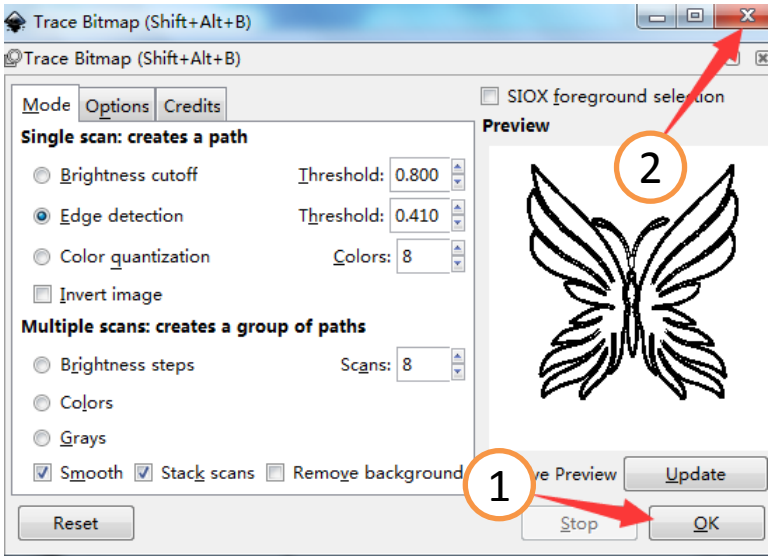


Set OPTION,

Smooth Coners must be disabled!!

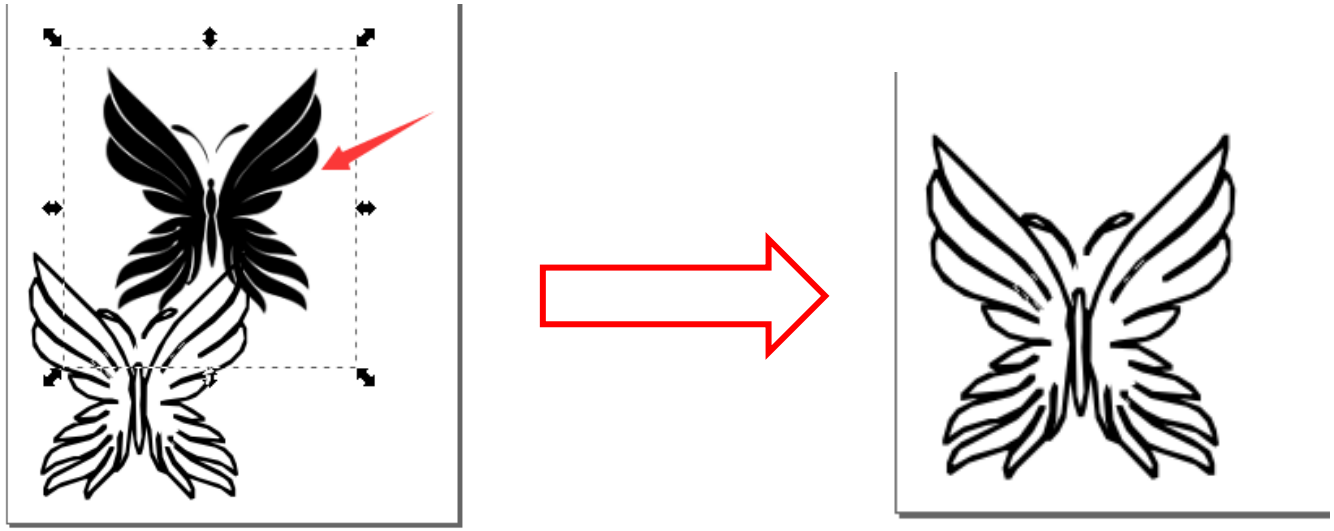
Zonestar

Generate gcode files using Laser Cut Gcode Generator



1. Click OK to generate the outer frame
2. Close the Track window

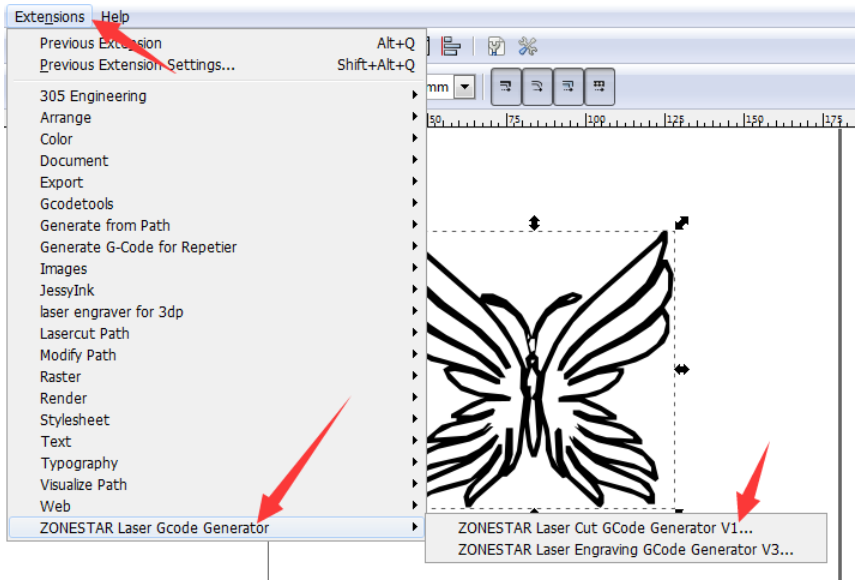
4. Remove the original image, leaving only the outer frame

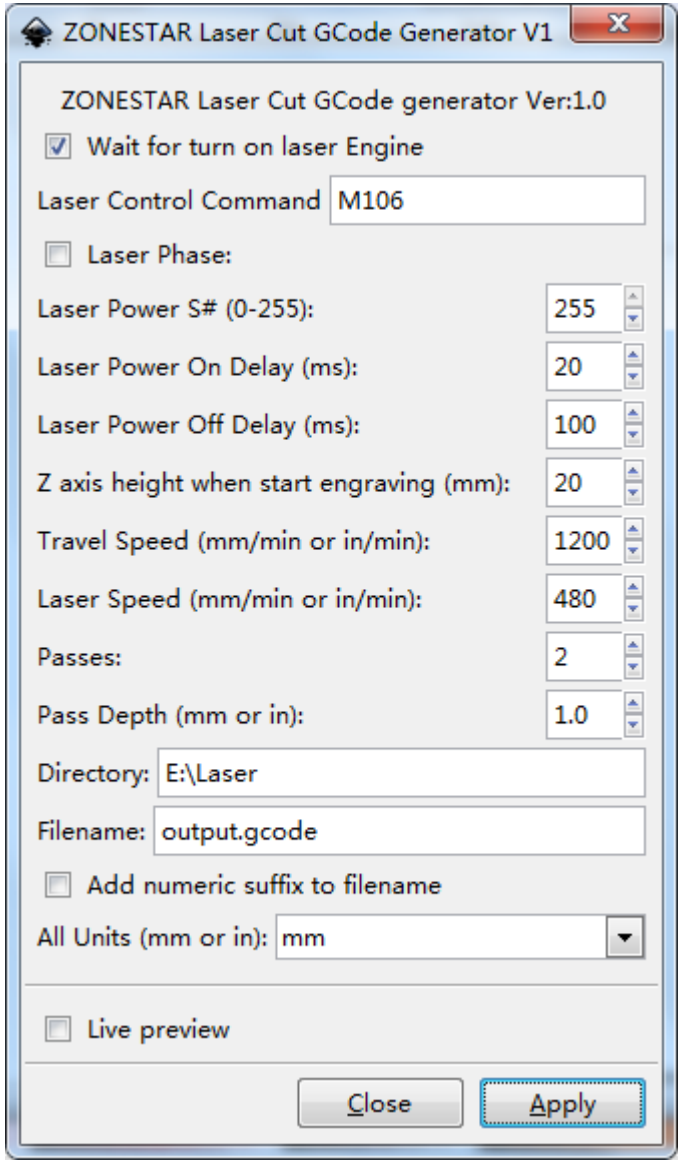


Zonestar

Generate gcode files using Laser Cut Gcode Generator

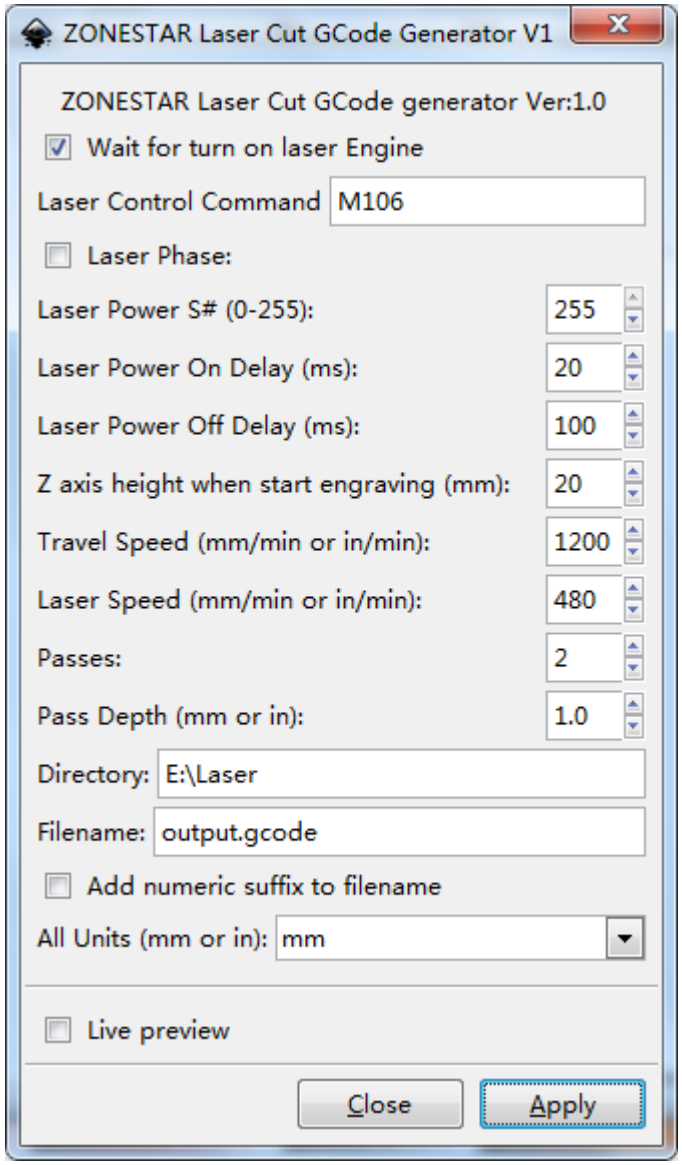
6. Click the Extensions menu, choose ***ZONESTAR Laser Gcode Generator>>ZONESTAR Laser Cut Gcode Generator*** and you will see the following window.





zonestar

Generate gcode files using Laser Cut Gcode Generator



Wait for turn on laser engine:
When this option is selected, it is generated. a pause command is automatically added to the gcode code to wait for you to turn on the laser engine, and a prompt on the LCD: "**Turn on Laser Power And Press knob**". It is recommended to ENABLE.

Laser control command : Laser control command, default is M106, it means use the TTL connect to FAN.

Laser Phase: The phase of the laser TTL signal. **DISABLE it for Wiring Mode A and ENABLE it for Wiring Mode B.**

Laser Power: Set the Max Laser Power.

Laser Power ON Delay: Set the delay time after the laser is turned on.

Laser Power OFF Delay: Set the delay time after the laser is turned off.

NOTE: The purpose of Laser power ON/OFF Delay is to reduce the "tailing" phenomenon.

Z axis height when start engraving: Sets the height at which the Z axis should rise before starts, according to the thickness of the object being carved.

Travel Speed: The XY move speed without engraving.

Laser Speed: The XY move speed when engraving.

Passes: Engraving line width.

Pass Depth: depth of engraving.

Dirctory: the saved directory of the generated gcode file.

Filename: the filename of the generated gcode file.

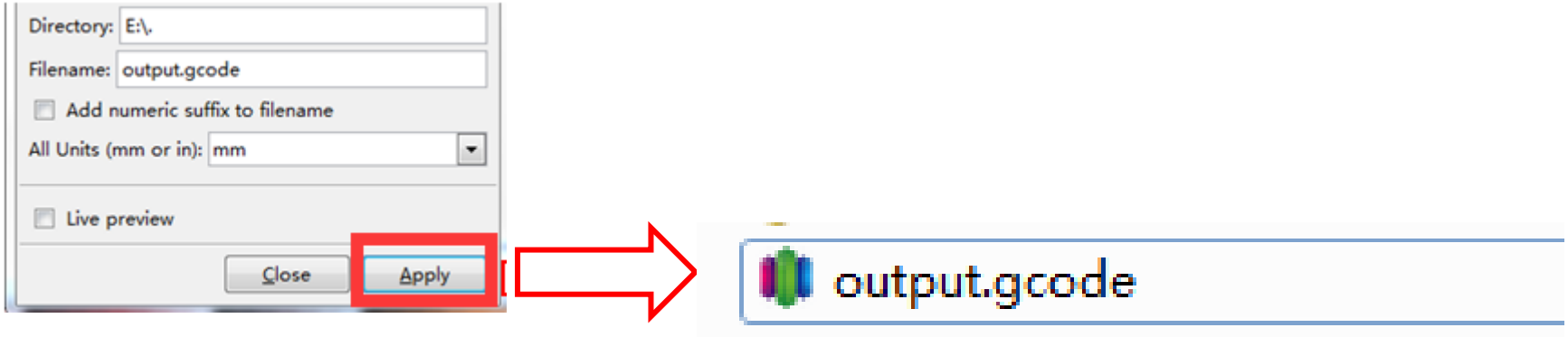
Add Numeric suffix to filename: automatically increate a number at the generated file name.

All units: unit of size

zonestar

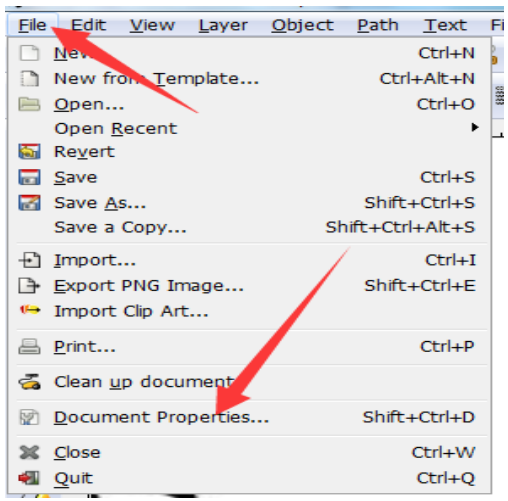
Generate gcode files using Laser Cut Gcode Generator

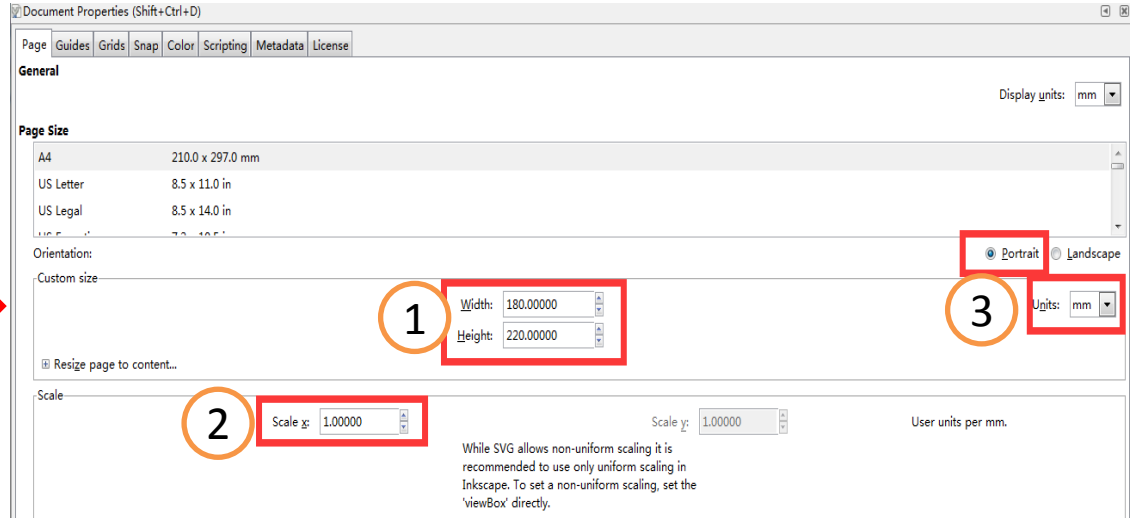
7. Click Apply. After a few seconds, you can find the **output.gcode** (the filename is depended on you filled in) file in the specified directory. You can copy this file to the SD card or directly “print it” by using the Host software.

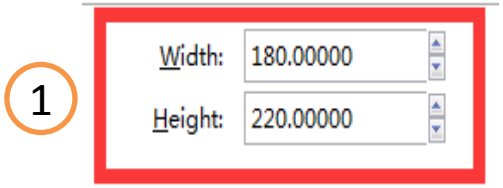


Generate gcode files By using Laser Engraving Gcode Generator

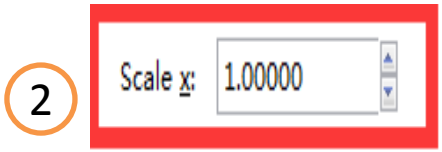
1. Settings



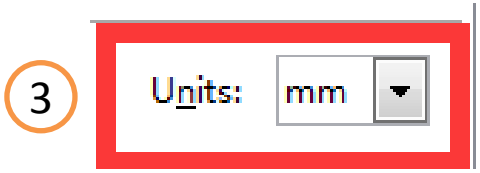




The width and height of the page are according to the actual engratable size, usually it is 40 to 50 mm smaller than the printer's building size in the X-axis and the length is the same as the Y axis. Anyway, it depends how to install the laser engine.



Set the scale to 1. If the actual size of the engraving is slightly different from the original size, you can modify this parameter to fine-tuning

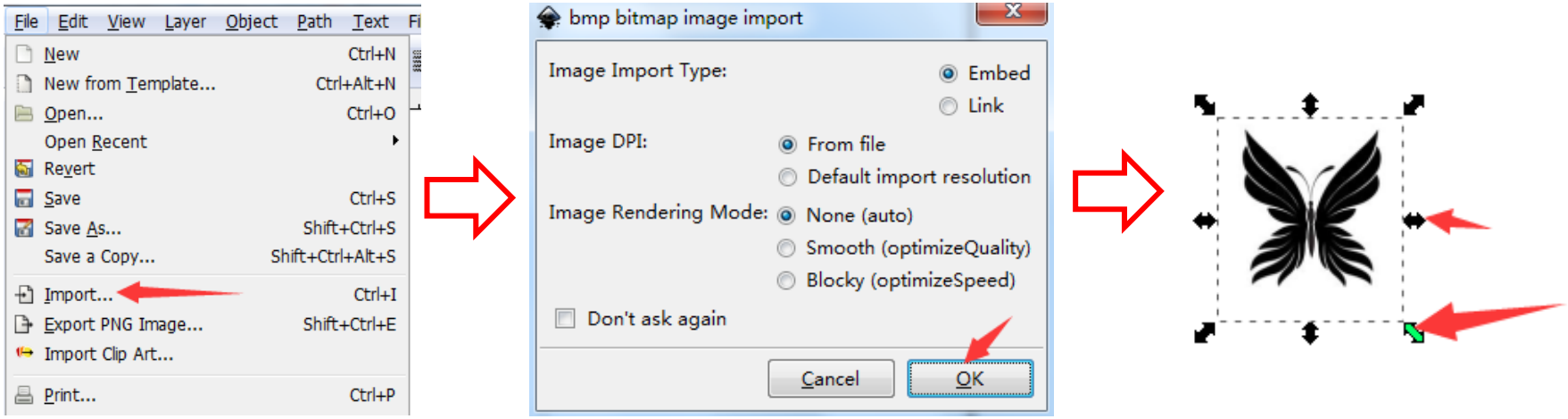


Set the size unit to mm.

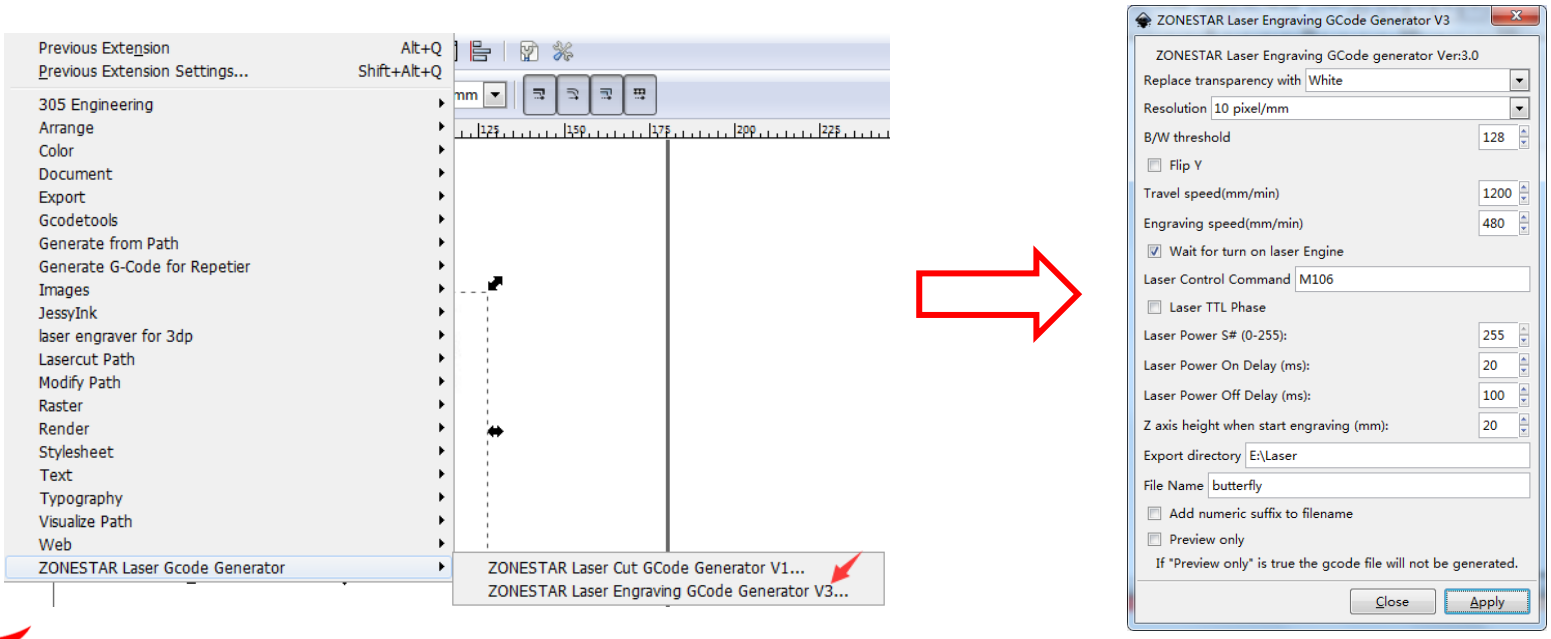
Zonestar

Generate gcode files By using Laser Engraving Gcode Generator

2. Import the image and zoom and place it in the position to be engraved.

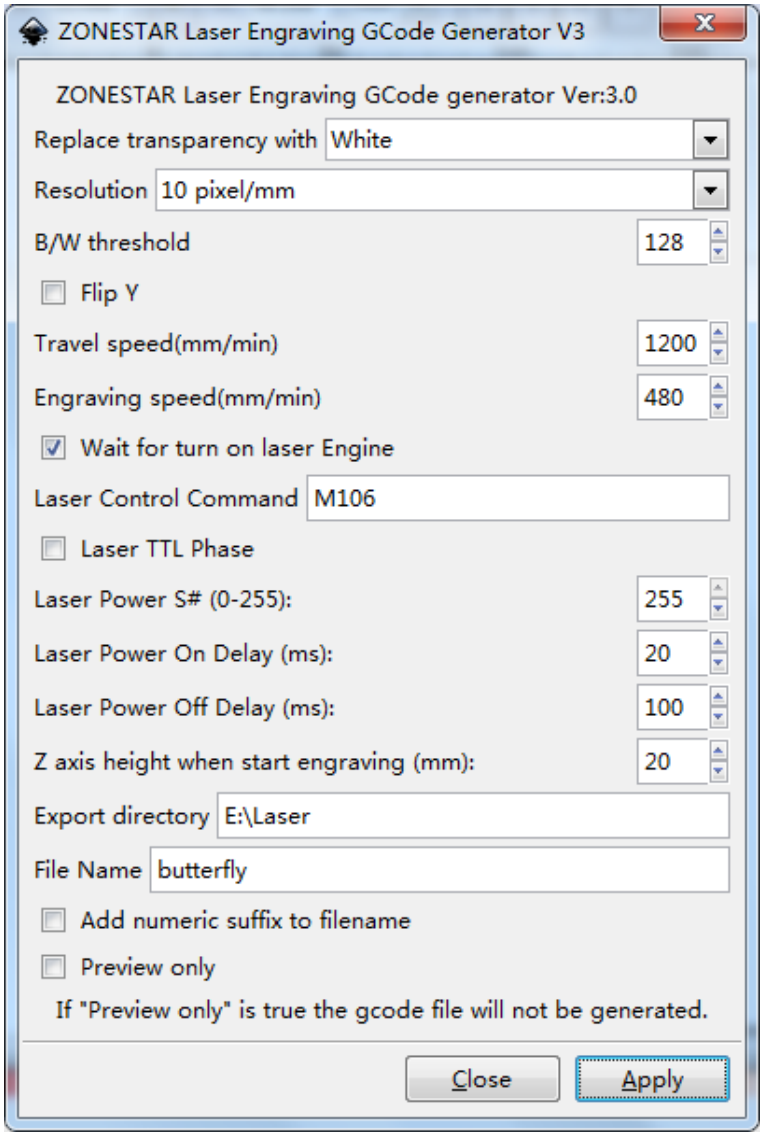


3. Click the Extensions menu, choose ***ZONESTAR Laser Gcode Generator>>ZONESTAR Laser Engraving Gcode Generator*** and you will see the following window.



zonestar

Generate gcode files By using Laser Engraving Gcode Generator

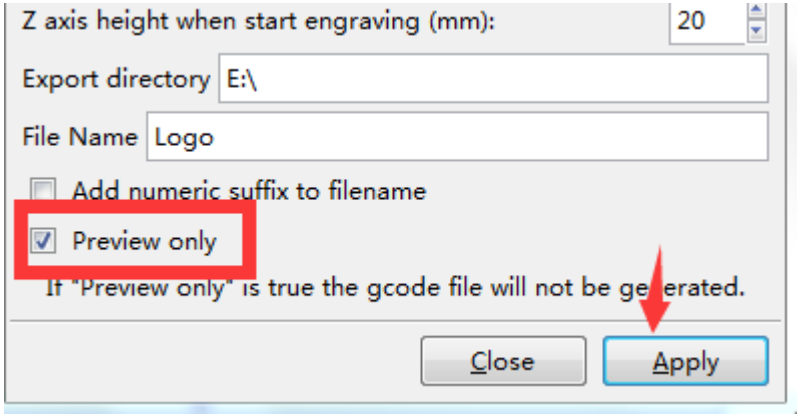


Replace transparency with: Fill the blank area of import picture by white(NO engraving) or black(engraving)**.**
Resolution: Set the engraving resolution.
B/W threshold: Threshold for grayscale image conversion (points with a picture brightness less than this value will be discarded).
Flip Y: Mirror Y
Travel Speed: The XY move speed without engraving.
Laser Speed: The XY move speed when engraving.
Wait for turn on laser engine:
When this option is selected, it is generated. a pause command is automatically added to the gcode code to wait for you to turn on the laser engine, and a prompt on the LCD: "**Turn on Laser Power And Press knob**". It is recommended to ENABLE.
Laser control command : Laser control command,
default is M106, it means use the TTL connect to FAN.
Laser Phase: The phase of the laser TTL signal. **DISABLE it for Wiring Mode A and ENABLE it for Wiring Mode B.**
Laser Power ON Delay: Set the delay time after the laser turned on.
Laser Power OFF Delay: Set the delay time after the laser turned off.
NOTE: The purpose of Laser power ON/OFF Delay is to reduce the "tailing" phenomenon.
Z axis height when start engraving: Sets the height at which the Z axis should rise before starts, according to the thickness of the object being carved.
Dirctory: the saved directory of the generated gcode file.
Filename: the filename of the generated gcode file.
Add Numeric suffix to filename: automatically increate a number at the generated file name.
Preview only: only generate a priview picture.

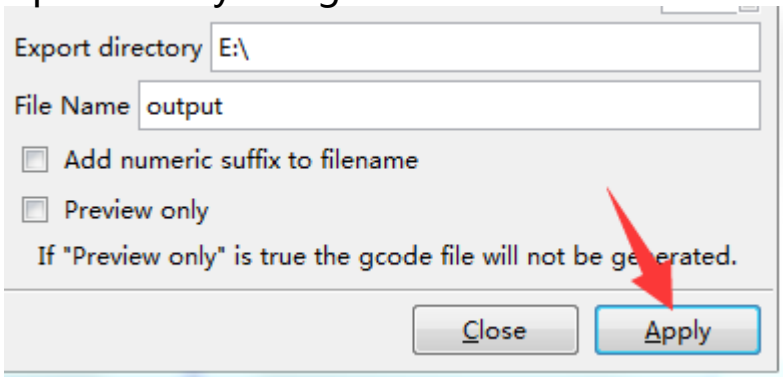
Zonestar

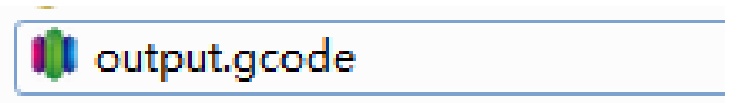
Generate gcode files By using Laser Engraving Gcode Generator

4. **Resolution** is the number of lines per millimeter of engraving when engraving. The higher the value, the finer the engraving, but the slower the speed. When the image you import is pure black and white (gray or color), you can set a **B/W threshold** and check "Preview only", then click apply, you can find a "xxxx_Preview.png" in your Export directory, check this image file to confirm if the parameters are set correctly.



5. Click Apply. After a few minutes, you can find the **output.gcode** (the filename is depended on you filled in) file in the specified directory. You can copy this file to the SD card or directly "print it" by using the Host software.





Enjoy it!