Contents

1,	Con	necting Power and Computers	3
	1),	Connecting Power	3
	2)、	Connecting Computers	4
2,	Insta	all drivers	4
3,	Pre-c	configurate the software and test	6
	1),	Connect the software	7
	2),	Test	10
4、	Print	t Models	12
	1),	Machine Adjustment	12
	2),	Generate STL file to GCODE	17
	3),	Printing	22
	4),	Off-line printing	22
We	bsites	S	23

Preparation before use:

When you received our 3d printer and open the carton, please do simple assembly first:

Put the assembled Y axis to sheet metal frame,

Tighten the nuts marked in red circle, ATTENTION, the threaded rod at red circle 1

and 2 must be very close to the inner part of the slot



Then connect the wire to the PCB, we have stick the marks on the wire to demonstrate which port on the PCB the wire should be connected

You need connect 4 sets of wire, the Y motor, the Y stop, the BTEMP, and the HOTBED.

PS: when you connect the Y motor, the order of the wire should be green, black, blue, red, from up to down, refer to the below pictures



1、 Connecting Power and Computers

1), Connecting Power

Connecting the plug to the socket



2), Connecting Computers

Connecting the machine and the computer with the USB wire,



2、Install drivers

Please copy the files in the SD cards to the computer for back-ups. Then could delete the



files, SD card would speciallized for off-line printing.

After successful connecting, computer will let us install the new hardware, cancel please.

If your computer system is XP, please follows the below steps:

STEP1- please install the DOTNETFX35 first, shut off the internet, the speed could be faster.

(PS: Win 7 or aboved version has that software)

STEP2- Find the automatically-installment program, double click setup to install the driver according to the wizard

PS: If the automatically-installment program don't work under Win 7 or above version, please install the driver manually! The methods are all the same.

Check whether the driver is installed successfully.

Take the USB out of the computer, and put it in again, the system will let install the new hardware again,

A: If you install the driver automatically, click next, system will install automatically, click OK at last.

B: If you do it manually, choose" Install from the list or specific location", then choose the catalogue of the driver, click install at last. (The same method as common driver manual installment).

Right click "My Computer", choose "Management", click "Device

Management", and refer to below pictures"



Find "Port", you install the计算机管理driver successfully.Pleasekindly keep your PORT number

(Every computer might have different port number).



3 Note: Pre-configurate the software and test

In the SD card, there is not only our company's own software, we also

supply the CURA.

1), Connect the software

Double click Han-bot 3d printing system.exe, Our printing system will

come out.:





Click it, you will see the window below,

Setting the CONNECTING first, still remember your port? Use yours. Suggested that set other parameter as the picture below, especially the Baud Rate must be

Printer Settings					
Printer: HanBot	×				
Connection Printer Printer Shape Advanced					
Port:	COM121 🖌 🖌 Refresh Ports				
Baud Rate:	115200				
Stop Bits:	1				
Parity:	None 💙				
Transfer Protocol:	Autodetect				
Receive Cache Size:	63				
 Receive Cache Size: 63 From Arduino 1 on the receiving cache was reduced from 127 to 63 bytes! Use Ping-Pong Communication (Send only after ok) The printer settings always correspond to the selected printer at the top. They are stored with every OK or apply. To create a new printer, just enter a new printer name and press apply. The new printer starts with the last settings selected. Delete This Printer Setting 					
OK Apply Cancel					

Setting the PRINTER, Attention please, the temperature is important, ABS, please set 230°C and 110°C, if you print PLA, please set 185°C and 55°C to 80°C,(We use ABS for this manual, so our temperature is 230 and 110°C.)

Setting the PRINTER CONFIGURATION. Our PRUSAI i3 build size is 200*200*200MM, so change the number to 200 in the market form, the other could be the same as below picture,

Printer Settings					
Printer: HanBot					
Connection Printer Printer Shape Advanced					
Printer Type: Classic Printer 🖍					
Home X: Min 💙 Home Y: Min 💙 Home Z: Min 💙					
X Min O X Max 200 Bed Left: O Y Min O Y Max 200 Bed Front: O					
Print Area Width: 200 mm					
Print Area Depth: 200 mm					
Print Area Height 200 mm					
The min and max values define the possible range of extruder coordinates. These coordinates can be negative and outside the print bed. Bed left/front define the coordinates where the printbed itself starts. By changing the min/max values you can even move the origin in the center of the print bed, if supported by firmware.					
OK Apply Cancel					

Then, click OK

PS: In the "Repetier General Settings", the workdirectory must be English, otherwise the GCODE cannot be generated. Refer to below picture

U 1 1 1	
forkalrectory.	sers(Administrator(AppLata)Local(hepetierhost Browse
🗖 Log Session	
In the work dire logging is enabl there will be a program will del	ctory, the logfile and temporary files for slicing are stored. If ed, file repetier.log after the program is closed. Starting the ete the
Behavi our	
🦳 Disable Quali	ty Reduction
FVI	
WI 🕅 Reduce Toolba	arSize

2), Test



Test the Moving control and the end-stop

Click "Homing" button \bigwedge , After that, move the X/Y/Z axis to check the movement.

 * - X homing, Y/Z is the same. After click, the axis will home.

After click, the 3 axis will home at the same time.

PS: If the machine is not homed, it could only move in Unilateral area; if you are not familiar with the machine, the move distance could not too far.

Check the temperature-test and heat system (Extrusion and heat bed)

If not heated, the number of temperature will be the same as ambient.





Check the fan

Adjust the fan output to 100%, then click **Fan**, check whether the fan works well.

PS: When heat the bed, shut off the fan.

4、 Print Models

1), Machine Adjustment

Actually, every Prusa I3 is well adjusted. But during the shipping, there could be some error because of the hit or other aspects. So before printing, please check whether the heat bed is flat or not again and whether the height between the heat bed and nozzle is OK.

Choose 4 corners of the heat bed, choose one first, Click

, homing Z axis,

check whether the distance between the heat bed the nozzle is almost the same as the below picture,



If the distance of the four corners is almost the same as reference, and there is not big difference, we don't need to adjust the machine.



But, If the distance is too large as the below picture

Please loose the Z end-stop holder first, and lower down the holder a little bit, click



again, check the distance, if still too large, lower down again until the distance is almost the same as the reference picture, then fix the holder.





If the distance is too small as the below picture:

Please loose the Z end-stop holder first and move up the holder a little bit, click



again, check the distance, if still too small, move up again until the distance is almost the same as the reference picture, then fix the holder.



After adjust one corner, don't forget to adjust the other 3 corners, during adjust the other 3 corners, there is no need to adjust the Z end-stop, we could set the screws on the heat bed, if the distance is too large, use the needle-nose pliers the fix the nut,, use M2.5 hex key to loosen the screw counterclockwise as the below picture, when you are doing that, check the distance, reach to the suggested distance is OK



If the distance is too small, we could use the needle-nose pliers the fix the nut, use M2.5 hex key to fix the screw clockwise as the below picture, when you are doing that, check the distance, reach to the suggested distance is OK



Here we have a good printing experience to share with everyone:

After the adjustment, there will some deviation; we have good and easy method to solve the problem:

When machine is printing the first layer, if the distance is too large, we could twist the 2 couplings counterclockwise simultaneously a little bit, until the distance reach to the reference,



if the distance is too small, we could twist the 2 couplings clockwise simultaneously a little bit, until the distance reach to the reference,



2), Generate STL file to GCODE

Our 3d printer support STL and Gcode file. And the it need the GCODE at last, the quality of the GCODE will influence the precision. To generate the high-quality GCODE, please handle the slicing configuration well.

If you have already turned on the software, please shut it down.



Find "Slicing Configuration", open it, double click" Double click and run", then press "Enter" button, then the slicing configuration well be set as our company's default. You can also change the settings according to your own demand.

Load Model

Open the software, in	Object Placement	, click	Add Object	, choose the
CTI was want to mint				

STL you want to print.





Models Mangement

You could see "Translation", "Rotation", "Scale" in the right part of the software,

Tra	inslation			
X	Y		Z	
Sca	ale			
X	Y		z	
			L	
1	Lock Aspec	t Ratio		
Ro	tation		_	
Х	Y		Ζ	

"Translation"- change the position

"Scale"- Change the size

"Rotation"- the angle of the model.

There are "Center Objects", "Copy Objects" buttons, in the down part, you could

try these functions.

Generating Gcode

If you print ABS, choose DRT2(ABS), If

PLA, choose DRT2(PLA)

物体放置	代码生成器(代码	编辑器	手动控制
	Sli	c3r	
○ 激活			☆ 配置
打印设置:	DRT2	*	
打印机设置:	DRT2	*	
原料设置:			
挤出头 1:	DRT2(ABS)	*	
挤出头 2:	DRT2(ABS) DRT2(PLA)		献 设置命令行
挤出头 3:	DRT2(ABS)	~	TO KENPETI
	co	de	
▶ 生质	战代码用 Slic3r		中止代码生成

If you want to change the settings, please refer to below pictures after click,

💋 Slic3r			
File Plater Window Help			
Plater Print Settings Filament Se	ttings Printer Settings		
drt 🔹 🗐 🥥	Layer height	\frown	
Layers and perimeters	Layer height:	0.3	mm
Infill	First layer height:	0.2	mm or %
 Skirt and brim Support material Notes Output options Multiple Extruders Advanced 	Vertical shells Perimeters (minimum): Randomize starting points: Generate extra perimeters when needed:	2	A V
	Horizontal shells Solid layers:	Top: 3	Bottom: 3

Layer height is suggested 0.3mm, first layer height 0.2mm

Speed for print moves					
Perimeters:	20	mm/s			
Small perimeters:	20	mm/s or %			
External perimeters:	40	mm/s or %			
Infill:	60	mm/s			
Solid infill:	60	mm/s or %			
Top solid infill:	50	mm/s or %			
Support material:	30	mm/s			
Bridges:	60	mm/s			
Gap fill:	20	mm/s			

Perimeters speed suggested 20, small perimeters suggested 20, external perimeters suggested 40.

Modifiers		
First layer speed:	30%	mm/s or %

First layer speed suggested 30%, if too fast, the parts could be warping.

💋 Slic3r			
File Plater Window Help			
Plater Print Settings Filament Set	tings Printer Settings		
drt 🔻 🗎 🥥	Filament	\frown	
Sector Filament	Diameter:	1.75 hm	
Cooling	Extrusion multiplier:	1	
	Temperature (°C)		
	Extruder:	First layer: 230	Other layers: 230
	Bed:	Krst layer: 110	◆ Other layers: 110
	•	III	•
Version 0.9.8 - Remember to check	for updates at http://slic3r.org/		

Filament diameter 1.75, extrusion multiplier 1, extruder temperature 230, bed

110(because we are printing ABS)

💋 Slic3r				×		
File Plater Window Help						
Plater Print Settings Filament Set	tings Printer Settings					
drt 🔹 📄 🤤	Size			Â		
General	Nozzle diameter:	0.4	mm			
Sustom G-code	Position (for multi-extruder printe	rs)				
	Extruder offset:	х: 0 у: 0	mm			
	Retraction					
	Length:	1	mm (zero to disable)	-		
	Lift Z:	0	mm			
	Speed:	30	mm/s			
	Extra length on restart:	0	mm			
	Minimum travel after retraction:	2	mm			
Retraction when tool is disabled (advanced settings for multi-extruder setups)						
	Length:	10	mm (zero to disable)	-		
Version 0.9.8 - Remember to check f	or updates at http://slic3r.org/					

Nozzle diameter 0.4

After every change, we must click		to save, remember the save name.
-----------------------------------	--	----------------------------------

Slic3r			
O Active			🐞 Configure
Print Settings:	DRT	*	
Printer Settings:	DRT	*	
Filament Settings:			
	DPT		
Extruder 1:	DEI	×	
Extruder 1: Extruder 2:	pangu	×	

Choose the saved configuration as the above picture,(I save the name as DRT) Now your setting is done.

Click Active then Click	with Slic3r
Gcode is being generated as the right picture	代码生成信息 生成器 Slic3r 奇作: Slicing SL file 持续时间: 0:23 □ 生成代码后开始打印
Gcode is generated	

3), Printing

After connecting the machine with computer, click ^{Run Job}, when the temperature rise to the setted, the machine will work.

PS: The first layer is very important, it is better that leave the machine after its successful printing.

4), Off-line printing

Off-line printing is better, especially for big models costing more time. After the GCODE is generated, click "Save" button and save it to SD card, the save

	物体	放置	代码生	É成器/	代目	编辑器	手动	加控制
B		≪ 40	610	CI	G-Code		•	
	1;	Floo	r = 35	50				
1	2; 3	gene	rated	by Sl	Lic3r	0.9.101	on	2014-02
	4;	laye	r_heig	ght =	0.2			
	5;	peri	meters	3 = 2				

name could only be in English or Arabic numbers After saved, please refer to the "LCD Control Systems" for more details.

Websites

Any question, please visit our websites.

- Website: <u>http://www.han-bot.com/</u>
- Forum: <u>http://www.han-bot.com/bbs/forum.php?mod=forumdisplay&fid=36</u>
- Alibaba: <u>http://han-bot.en.alibaba.com/</u>