

# User manual: Orca



User manual: Orca 0.4x

Version: 1.2

March 2014



### Content

Con	tent
1.	Safety
2.	Specifications
Ρ	rinting4
D	imensions
T	emperature
3.	FFF 3D-print technology
4.	Setting up the Orca
5.	Installing the software7
	Step 1, driver install
	Step 2, Repetier install
6.	Printer Settings
7.	Installing Print Profile
8.	Prepare G-code for printing
9.	Printing
10.	Calibration of the axis
11.	Replacing material 23



# 1. Safety

	Safety alert	symbols precede each safety message in this manual. These symbols
$\dot{}$	indicate pot	ential safety hazards that could harm you or others or cause product or
20	property da	mage.
	Warning:	The Orca generates high temperatures. Do not reach inside the machine
		when operating. Always allow the Orca to cool down first.
	Warning:	The Orca includes moving parts. Never reach the Orca when the machine
20		is in operation.
	Warning:	Do not leave the Orca unattended during operation.
- <del></del>		
	Caution:	Only use the power supply provided with your Orca.
$\overline{\mathbf{A}}$	Caution:	We recommend only to use materials supplied by Orcabot. Other
<u> </u>		materials can have unexpected properties with can influence the quality
		or worse.
	Caution:	In case of emergency disconnect power supply from wall socket.
	Caution:	Orca melts plastic during printing. Plastic odors are emitted during this
		operation. Make sure to set up the Orca in a well-ventilated area.



# 2. Specifications

### Printing

Print technology:	Fused Filament Fabrication (FFF)
Layer resolution:	100 μm
Print resolution:	0,02mm
Nozzle diameter:	0,35mm
Filament:	1,75mm Orcabot filament
Heated bed:	Yes

### Dimensions

Build volume:	250x220x190mm
Frame dimensions:	380x280x420mm
Total weight:	9,4 kg.

### Temperature

Ambient operating temperature:	10°C - 40 °C
Storage temperature:	0°C - 50 °C



### 3. FFF 3D-print technology

Fused filament fabrication (FFF) is an additive manufacturing technology commonly used for modeling, prototyping, and production applications.

FFF works on an "additive" principle by laying down material in layers; a plastic filament or metal wire is unwound from a coil and supplies material to produce a part.

The technology was developed by S. Scott Crump in the late 1980s and was commercialized in 1990.



### 4. Setting up the Orca

After building or unpacking the Orca, the next step is setting up the printer. A few things are important:

- The Orca must be placed on a smooth and level surface
- There must be an electric socket, recommended is a socket with an on/off switch
- It is recommended to choose a room or place which has a decent ventilation

The Orca package contains the Orca printer, a USB cable and two power supplies. The USB cable is for connecting the printer to a computer. The two power supplies are for:

- 1. Heated bed
- 2. Electronics etc.

Before testing and connecting the printer to the computer, choose a solid table or desk, to put the printer on. Switch the machine on by switching on the two power supplies (the Orca itself doesn't have an On/Off switch) and connect the Orca with a computer via the USB cable.

The computer will detect a new USB device on one of the COM ports.



### 5. Installing the software

Windows only, there is also an Apple OS package.

On the Orcabot website you can download the driver and software to control your Orca 3D printer.

#### Step 1, driver install

Open the folder with all the unpacked files in it. First open "1-DRIVER-INSTALL". Then open the file "FTDI\_win7\_XP\_VISTA-2K\_x64\_x32". This folder contains the file "CDM20802\_Setup", run this file twice. The second time a command screen will pop up.

#### **Step 2, Repetier install**

Open the folder "2-REPETIER-INSTALL", select the "setupRepetierHost\_0\_95" and install it.

First, choose your preferred language for the installation.



The following dialog will appear:

Click: "Next"



If you agree with the license agreement, select "I accept the agreement" and click "Next".





Select the destination folder and click "Next".

Select the start menu folder and click "Next".

Select the checkbox to create a desktop icon and click "Next".



😽 Se	tup - Repetier-Host	_ <b>D</b> X
R	eady to Install Setup is now ready to begin installing Repetier-Host on your computer.	Ð
	Click Install to continue with the installation, or click Back if you want to revi change any settings.	ew or
	Destination location: C:\Program Files\Repetier-Host Start Menu folder: Repetier-Host	*
	Additional tasks: Additional icons: Create a desktop icon	
	٠	
	< <u>B</u> ack Install	Cancel

Click "Install" to proceed with the installation.



Finally click "Finish" to finish the installation.



### 6. Printer Settings

Start Repetier Host and select "Printer settings" in the right top corner.

nnect Load Save Job Run Job Kill Job SD Card Toggle Log Hide Filament	Printer Settings	imergency S
Temperature Curve	Olject Hademerk Store (G-Code Editor Menual Control	
	Name Mesh	Collision
	Rotation X Y Z	
	Cut Objects	
	Poston	
	Admuth	
v t ,		
✓		
*		
26:34.618 OpenGL version:2.1.0 - Build 8.15.10.2182	i III uwartog t <u>Cl</u> copy	
26:34.619 OpenGL extensions:GL_EXT_blend_minmax GL_EXT_blend_s 26:34.619 OpenGL renderer:Mobile Intel(R) 4 Series Express Chip	btract GL_EXT_blend_color GL_EXT_abgr GL_EXT_texture3D GL_EXT_clip_volume_hint GL_EXT_compile set Family	d_vertex
26:34.650 Using fast VBOs for rendering is possible		

Please enter the following settings in the four TAB's and after that press "Apply" and "OK".

Printer Settings	a forma situa difort Topping Habilianas Deviced	Printer Settings	a stran SFG	r Topping Hakflamet Des Text
Printer: ORCA 0.4	4x * 💼 -	Printer: ORCA 0.4x		• #
Connection Printer Print	ter Shape Advanced	Connection Printer Printer Shape	Advanced	
Connector: Serial	Connection	Travel Feed Rate:	4800	[mm/min]
		Z-Axis Feed Rate:	100	[mm/min]
Port:	COM4   Refresh Ports	Default Extruder Temperature:	220	<b>2</b> °
Baud Rate:	250000 -	Default Heated Bed Temperature:	55	<b>*</b>
Transfer Protocol:	Autodetect 👻	Number of Extruder:	1 4	
Reset on Connect	Disabled	Check Extruder & Bed Temperat	ture	
Reset on Emergency	Send emergency command and reconnect	Remove temperature requests fr	om Log	
Receive Cache Size:	127	Check every 3 seconds.		
	From Arduino 1 on the receiving cache was reduced from 127 to 63 bytes!	Park Position: Y		714- 0
Use Ping-Pong Con	nmunication (Send only after ok)	E Cand CTAte added dialar	. 0	2-win U [min]
The printer settings alw	ravs correspond to the selected printer at the top. They	Go to Park Position after Job /Kill		
are stored with every O	K or apply. To create a new printer, just enter a new	Disable Extruder after Job /Kill		
selected.	appy, the new planter states with the last setungs	Disable Heated Red after Joh/Kil	(	
		Disable Motors after Job/Kil		
		Add to come Disting Two	19/1	
		Aud to comp. Plinting Time a	[20]	
	OK Apply Cancel		ОК	Apply Cancel
	See 1999			



Printer Settings	Printer Settings
Pinter: ORCA 0.4k  Connection Pinter [Pinter Shape] Advanced	Printer: ORCA 0.4x
Printer Type: Classic Printer	Post Slice Filter
Home X: Min V Home Y: Min V Home Z: Min V	Filter Path and Parameter:
XMn       0       X Max       250       Bed Left:       0         YMn       0       Y Max       220       Bed Front:       0         Print Area Width:       250       mm         Print Area Depth:       220       mm         Print Area Height:       190       mm         The min and max values define the possible range of extruder coordinates. These coordinates on the print bed. Bid eff. front define the coordinates where the printibed Iself starts. By charging the min /max values you can even move the origin in the certer of the print bed. if aupported by firmware.	Run Filter after every Sice You can run a filter program after each slicing action. The filter will be run on the G-Code produced by the after. Use iffin and flout to insert the input and output filenames as parameter.
OK Apply Cancel	OK Apply Cancel



### 7. Installing Print Profile

Next step is to install the Print Profiles in Repetier Host. These files include the layer height and settings for printing a specific material like: PLA, ABS, Nylon etc..

You can download the Repetier Print Profiles from the Orcabot website.

Go to the TAB Slicer and click on the button "Manager".

View Temperature Curve	Object Placement Silcer G-Code Editor Manual Control	
	Slice with Slic3r	Kill Slicing
	Sicer Sicir	• 0% Manager
		Configure
K /	Print Setting: Silc3r 0.20L-PLA met support	•
	Printer Settings Silc3r 0.20L-PLA	•
	Filament settings:	
	Edruder 1: Sic3r 0.20L-PLA	•
	Extruder 2: Sac3r 0.20L+PLA	•
	The Control of the Cathoon	
	Copy Pirt Settings to Ovende	
v t ,	Enable Suppor	
▼.	Enable Cooling	
	Layer Height 0.2 mm	
₩.	Print Density	2014

Select "Slic3r" at the left side of the dialog and click "Apply".

Slicer Configurations	Setup
Slic3r Skeinforge	Slic3r Configuration: Slic3r Slic3r Slic3r Configuration Directory
	Browse Leave blank to use guessed location.
	Slic3r Executable
	Browse
	Slic3r Version
	0.9.9 or higher 💌
	With these settings, you are able to use different versions of Slic3r. You can get new versions from http://www.slic3r.org The minimum version required is 0.9.0. Leave all fields blank to simply use the bundled version.
	Apply Reset
Slicer Slic3r	
Name	
Add Slicer	



Next click "Configure".

Plater Print Settings Filament Settings Printer Settings		
Add Sciency   Handel Sciency   Hindel	Image: Solution of the second seco	) 🧼 S 35:
Drag your objects here	Printer: - default - Export C Export S	i-code.
	Info Size: Volume: Facets: Materials: Manifold:	

Click "File"; "Load Config" and select the Orca Slic3r Profile.

e <u>P</u> later <u>W</u> indow <u>H</u> elp				
Load Config	Ctrl+L			
Export Config	Ctrl+E	🕼 🥥 🔉 Rotate	🖁 🖉 Scale 🎼 Spli	t 🧐 View 🎼 Setting
Quick Slice	Ctrl+U	Name	Cop Sca	Print settings:
Quick Slice and Save As	Ctrl+Alt+U			- default -
Repeat Last Quick Slice	Ctrl+Shift+U			Filament:
Slice to SV <u>G</u>	Ctrl+G			- default -
Repair STL file				Printer:
Combine multi-material STL files				- default -
Preferences				Street Cards
<u>Q</u> uit				Export G-code
		<	+	Export STL
		Info		
		Size:	Volume	:
		Facets:	Materia	ls:
		Manifold:		

Go to the TAB "Print settings" and save the settings.



ile <u>P</u> later <u>W</u> indow <u>H</u> elp				
Plater Print Settings Filament	Settings Printer Settings			
Slic3r 0,20L-PLA.ini 🔹 📄	Layer height			
Layers and perimeters	Layer height:	0.2	mm	
Infill	First layer height:	0.2	mm or %	
Speed Skirt and brim	Vertical shells			
Support material	Perimeters (minimum):	2		
Output options	Spiral vase:		V	
Advanced	Horizontal shells			
	Solid layers:	Top: 4	Bottom: 4	
	Quality (slower slicing)			
	Extra perimeters if needed:			
	Start perimeters at:	Concave	e points: Non-overhang points:	
	Detect thin walls:			

Go to the TAB "Filament settings" and save the settings.

Slic3r			
<u>File Plater Window H</u> elp			
Plater Print Settings Filament Setti	ngs Printer Settings		
Slic3r 0,20L-PLA.ini 🔹 🔚 🥥	Filament		
🚸 Filament	Diameter:	1.75	mm
Cooling	Extrusion multiplier:	1	
	Temperature (°C)		
	Extruder:	First layer	0 Other layers: 0
	Bed:	First layer	0 Other layers: 0
Version 1.0.0RC2 - Remember to check t	or updates at http://slic3r.org/		

Go to the TAB "Printer settings" and save the settings.



e <u>P</u> later <u>W</u> indow <u>H</u> elp			
ater Print Settings Filamen	t Settings Printer Settings		
ic3r 0,20L-PLA 🔹 👻	Size and coordinates		
General	Bed size:	x: 230 y: 200 mm	
Custom G-code	Print center:	x: 100 y: 100 mm	
Extruder 1	Z offset:	0 mm	
	Firmware		
	G-code flavor:	RepRap (Marlin/Sprinter/Repetier)	
	Use relative E distances:		
	Capabilities		
	Extruders:	1	
	Advanced		
	Use firmware retraction:		
	Vibration limit:	0 Hz	

Make sure you also select the new profiles in the Repetier drop down menu's.

Slice with Slic3r  Socie  Socie Soc	Kill Slicin     Q\$ Manager     @\$ Configure
Skc3r           %           Skc3r 0.20L-PLAmet support           tings           Skc3r 0.20L-PLAmet support	Old Manager     Old Manager     Old Configure
ng: Silic's 0.20L-PLA met support tings: Silic's 0.20L-PLA	@ Configure
rg: Sic3r 0.20L-PLA met support tings: Sic3r 0.20L-PLA	
tings Skc3r 0,20L-PLA	
settings:	
Sic3r 0,20L-PLA	
Sic3r 0.20L-PLA	
My Settings	
de Slic 3r Settings	
Copy Pint Settings to Overside	
le Support le Cooking	
<i>tt</i> : 0.2 mm	
tyO	205
der 3 verri Frab Frab	ker 3. My Settings verifie Sch Settings Capy Perz Settings to Oversite value Second verifies



### 8. Prepare G-code for printing

Before anything can be printed, a 3D file must be exported to or saved as a ".STL" file. These files are suitable for 3D printing. Make sure when saving the file, that the Z-axis is pointing in the right direction (UP).

Repetier-Host V0.95F	Lines - Michael Moder		- • • ×
File View Config Temperature Printer Tools Help U O O Connect Load Save Job Run Job Kill Job SD Card Toggle Log Hide Filament Show Travel		Printer Settings	Emergency Stop
3D View Temperature Curve	Object Placement   Slicer   G-Code Editor   Manual Control		
C			
• •	(Add Object)	Mesh	Collision
	Tanulation X Y Z and Action		
Sowning OCommands OHfes OWarman Offen OACK OAdsSord @ Daring () 15356154615 OpenCi restantil.1.0 - Build 5.15.10.2121 15356154(4) OpenCi restanting (LET Lined jummas GL_ET Lined jumm	Copy d_color GL_EXT_shgr GL_EXT_texture3D GL_EXT_clip_volume_hint	GL_EXT_compi	Led_vertex_(

In Repetier STL files can be loaded and will be shown in 3D. Just click on the button "Add object" in the TAB Object Placement and choose the STL file or files. Everytime when a new object is loaded, it will be placed automatically in the screen and will be show in the list of STL objects. It is also possible tot manually move them with the mouse, or use the button: "Center object".



There is also the possibility to scale the models or place/rotate them with coordinates.



Next step is to Slice the Object. Go to the Slicer TAB and click "Slice with Slic3r". This can take some time.

Repetier-Host V0.95F - ListViewItem: (orca-043-1721_spool-top.stl)	on Monad Red	-	- 0 ×
File View Config Temperature Printer Tools Help			A0 53
Concept Land Streetable Run lab. Kill lab. 50 Card Tanala Lan. Mida Element Show Toront			Mig Distance Consumers State
3D Vew Temperature Carve	Object Placement S	087 G-Code Editor Manual Control	Printer Settings Emergency stop
C			
U C		Slice with Slic3r	Kill Slicing
÷			
<b>₽</b>	Slicer: Slic3r		Ø <sup>0</sup> <sub>0</sub> Manager
			tot Configure
	Print Setting:	Silc3r 0.20L-PLA met support	•
	Printer Settings:	Silc3r 0,20L-PLA	•
	Filament settings:		=
	Edruder 1:	Silc3r 0,20L-PLA	•
	Extruder 2:	Silc3r 0.20L-PLA	Ψ
	Extruder 3:	My Settings	
	Override Slic3r Se	ttings	
7	Copy Pri	int Settings to Override	
	Enable Support		
	Laver Height:	2	
	Infil Density		20%
	and borning		*
Show in Log: OCommands O Infos O Warnings O Errors O ACK O Auto Scroll  Clear Log  Cloar	у		
16:20:30.150 (Slic3r) => Compining infili 16:20:30.159 (Slic3r) => Infiling layers			^
16:28:31.216 <slic3r> =&gt; Generating support material 16:28:31.949 <slic3r> =&gt; Generating skirt</slic3r></slic3r>			
16:28:31.955 <slic3r> =&gt; Exporting G-code to C:\Users\tonniej.FRODIM-INT\AppData\Local</slic3r>	\RepetierHost\com	sposition.gcode	
16:20:32.627 (Slid3r) Done, process took o minutes and 3.532 Seconds 16:20:32.627 (Slid3r) Filament required: 2713.5mm (6.5cm3)			
Disconnected - Idle 116 EPS			•

After the slicing is done, the program will automatically change to the tab G-code.



In the object screen a blue "layered" object will show up (if not click on the "Show Filament" button). The dark blue line show the object layer by layer. The light blue lines show the Z-movements (no printing).

You can save the G-code by pressing the Save Job button on top of the screen.



### 9. Printing

It is always important to start with a clean Printbed. Degrease the Printbed with a towel and e.g. Acenton.

To start printing, the Orca must be switched on and connected to the computer via the USB cable. After that click on the button "Connect" in the top left corner.



The big "Cross" with the arrows are for manual control of the printer. In the corners you can find the home buttons (homeX, homeY, homez and homeALL).

Be carefull! Only use homeZ / HomeALL when the Z-is calibrated. Try some arrows and make some moves to make sure that the printer is online and correctly connected.





When the axis are not calibrated yet, please go to the chapter: "Calibration of the axis". Otherwise you can proceed this chapter.

The next step is to set the temperature for the Extruder and the Printbed. First fill in the requested temperature and after that click on the button "Heat Extruder" and "Heat Printbed". The temperature values in grey are the current temperatures.



Always wait untill the right temperature is reached before continuing!

For temperature settings, please go to <u>www.orcabot.com</u>.

If the temperature is reached, the printer is almost ready for printing, but first manually feed the extruder with some material.



Use the big wheel to feed the extruder and hold the material wire with the other hand, feed it until a good amount of material is flowing through the extruder underneath. After this, use a tweezer to remove the material.



#### Warning: the heater is very hot, so don't touch it with bare hands!

#### Warning: never touch the printer bed or extruder with bare hands!

Always use tweezers or pliers to remove prints from the bed. If the bed is touched with bare hands, it's possible that the bed lose its cohesion. If so, then degrease the bed with acetone.

If this is done, press "Run Job".



While printing, it's always very important to check the first layer and amount of printed material.

With this wheel on the Z-axis, the height of the bed can be changed (this is only useful for the first layer while printing).

Check the line of material. If it's too thin turn the wheel clockwise to raise the bed a bit. Do this until the lines are completely closed. If the lines are too thick and the material is really "sweeping up", then turn the wheel counter clockwise to lower the bed a bit.

After printing, the machine will automatically cool of and lower and move the bed away from the extruder head. Wait for the machine to cool off and then remove the printed object. Reminder, never touch the bed of extruder with bare hands. Use pliers or tweezers for removing parts (not manually, otherwise you need to degrease again).



### **10.** Calibration of the axis

The automatic homing of the bed and machine is done by opto sensors for the X-, Y- and Z-axis. These sensors have to be once calibrated before using the machine for the first time.



The homing sensor for the X-axis is located on the top of the machine. Adjust the opto flag (A) so it will move between the opto sensor (B).

The homing sensor for the Y-axis is located at the bottom of the bed (at the back side). Adjust the opto flag so it will move between the opto sensor.

The homing sensor for the Z-axis is located at the left front side of the machine. This homing sensor is very important!

First adjust the opto flag so it will move between the opto sensor, but do not cut the flag too short. Second step is to rotate the bolt (C) the move the opto sensor a bit up and down.

The final distance after homing between the nozzle and the bed should be 0,1 to 0,2 mm.

After the homing is set, we are going to calibrate the steps per mm. Therefore we open Repetier and connect with the Orca printer.



After that we go to the menu: "Config"; "Firmware EEPROM configuration".

leps per mm:	X:	32.05	Y:	32.05	Z:	960.00	E:	167.00
laximum feedrates [mm/s]:	X:	500.00	Y:	500.00	Z:	10.00	E:	100.00
aximum Acceleration [mm/s²]:	X:	9000	Y:	9000	Z:	100	E:	10000
coeleration:		1200.00						
etract Acceleration:		1200.00						
ID settings:	P:	22.20	l:	1.08	D:	114.00		
oming Offset:	X:	0	Y:	0	Z:	0		
Advanced Variables:								
Min feedrate [mm/s]		0.00		Maxin	num X	-Y jerk [mm/s	]	20.00
Min travel feedrate [mm/s]		0.00		Maxin	num Z	jerk [mm/s]		0.40
Minimum segment time [ms]		20000						

Write down the current Steps per mm values.

Now we are going to print a test file (cube), e.g. 20x20x20mm. When the cube is ready, measure the dimensions. Now go back to the "Firmware EEPROM configuration" menu and correct for the Steps per mm. Use the following formula:

$$S = N/R*C$$

- S = Steps per mm (new)
- N = Nominal dimension
- R = Real / measured dimension
- C = Current steps per mm (as written down in the beginning of this procedure)

Fill in the new values and press "Save to EEPROM".



### **11.** Replacing material

If the spool with material is (almost) empty, or when another material or color is wanted, it is time to replace the spool.

Switch on the printer and run Printer interface. Switch on the heater and wait till it has reached its temperature.



Try to leave min. 1 cm filament sticking out. After that, reverse the extruder by turning on the big gear and remove the old filament which was in the extruder. After that insert the new filament.

In case, filament runs out during a print, press "Pause" and unscrew the four bolts and remove the part in front of it. Use the wheel attached on the extruder to remove the material from the extruded. Then replace the spool of material with the new material and place wire through

the tube and also through the little hole on top of the extruded. Guide via the extruded wheel into the actual heater. Manually add some material and push it through the extruded until some material comes out underneath of it.

Place back the part with the four screws and tighten it until the material can be extruded by only using the wheel. It it's done correctly, the printer itself can extrude the material. After doing so, remove the leftovers from the nozzle underneath.