# COCOON create™

# 3D Printer

COCOON create™

Model Number: HE150272

## INSTRUCTION MANUAL

# Important

**Getting the most out of your 3D Printer:**

**Read the manual carefully**
It will help you get set up with ease

**Visit www.cocooncreate.com.au**
We've got additional support videos and guides

**Call After Sales Support - 1300 663 907**
Available Monday - Friday, 8:30am - 5:30pm AEST

**Email customercare@winplus.com.au**
We're always happy to help

# Warranty Note

If you experience any issues with this product, or it's performance is not what you had expected, please contact us at Winplus before returning the item to the store.

It is likely that we can resolve any problems for you via phone or email.

We can be reached on:

Phone: 1300 663 907 (Mon - Fri; 8:30am - 6:00pm EST)
Email: customercare@winplus.com.au
Web: www.winplus.com.au

# COCOON™
## create

## 3D Printer

# Warranty Details

The product is guaranteed to be free from defects in workmanship and parts for a period of 12 months from the date of purchase. Defects that occur within this warranty period, under normal use and care, will be repaired, replaced or refunded at our discretion. The benefits conferred by this warranty are in addition to all rights and remedies in respect of the product that the consumer has under the Competition and Consumer Act 2010 and similar state and territory laws.

Our goods come with guarantees that cannot be excluded under the Australian Consumer Law. You are entitled to a replacement or refund for a major failure and for compensation for any other reasonably foreseeable loss or damage. You are also entitled to have the goods repaired or replaced if the goods fail to be of acceptable quality and the failure does not amount to a major failure.

# Safety Instructions

Before you get started, please read these important safety instructions.

**CAUTION:** The Cocoon Create 3D Printer generates high temperatures and includes moving parts that can cause injury. Never reach inside of the Cocoon Create 3D Printer while it is in operation, and allow time for the printer to cool down after operation.

**HOT SURFACE**          **MOVING PARTS**

Vapours/fumes may be irritating at operating temperatures. Always use the Cocoon Create 3D Printer in an open, well ventilated area.

Do not leave the Cocoon Create 3D Printer unattended during operation

Contact with extruded material may cause burns. Wait for printed objects to cool before removing them from the build platform.

Always take care when using the Scraper. Never direct the Scraper towards your fingers.

- The printer must not be exposed to water or rain, or damage will occur.
- The printer is designed to be used with ambient temperature ranging 15°C - 30°C, and humidity ranging 20% - 50%. Operating outside these limits may result in low quality models.
- In case of emergency, turn off the 3D printer at the power outlet.
- It is recommended that you use eye protection when cleaning/sanding printed models to avoid small particles contacting eyes.

# Contents

# COCOON
*create*

# Parts List

1 x Cocoon Create 3D Printer (Control Box, Extruder Tower, Heated Print Bed)
1 x Filament Spool Holder
1 x Filament Spool Stand
1 x Power Cable

1 x USB Cable
1 x 4GB MicroSD Card
1 x Scraper
1 x PTFE Tube
1 x Hex Key Set
6 x Hex Bolts

2 x Cable Ties
1 x 10m PLA Filament
2 x Print Bed Mats
1 x Instruction Manual
1 x Getting Started Guide
1 x Warranty Card

# Technical Specifications

**Printing**

| | |
|---|---|
| Print Technology: | Fused Filament Fabrication (FFF) |
| Build Volume: | 200 x 200 x 180mm |
| Layer Resolution: | 0.1 – 0.4mm |
| Positioning Accuracy: | X 0.012mm |
| | Y 0.012mm |
| | Z 0.004mm |
| Extruder Quantity: | Single Extruder |
| Extruder Diameter: | 0.4mm |
| Print Speed: | 10mm/s - 70mm/s |
| Travel Speed: | 10mm/s - 70mm/s |
| Supported Print Materials: | Cocoon Create Branded PLA, ABS, PVA, HIPS, PLA Wood, PLA Flex, Conductive |

**Temperature**

| | |
|---|---|
| Ambient Operating Temperature: | 15°C – 30°C |
| Operational Extruder Temperature: | 170°C - 260°C |
| Operational Print Bed Temperature: | 30°C - 120°C |

**Software**

| | |
|---|---|
| Slicer Software: | Cura V14.07 |
| Cura Input Formats: | .STL, .OBJ, .DAE, .AMF |
| Cura Output Format: | .GCODE |
| Connectivity: | Micro SD Card, USB port (Expert users only) |

**Electrical**

| | |
|---|---|
| Input Rating: | 100-240V AC, 50/60Hz, 3.5A |

**Physical Dimensions**

| | |
|---|---|
| Printer Frame Dimensions: | 400 x 410 x 400mm |
| Weight: | 10KG |

**After Sales Support**
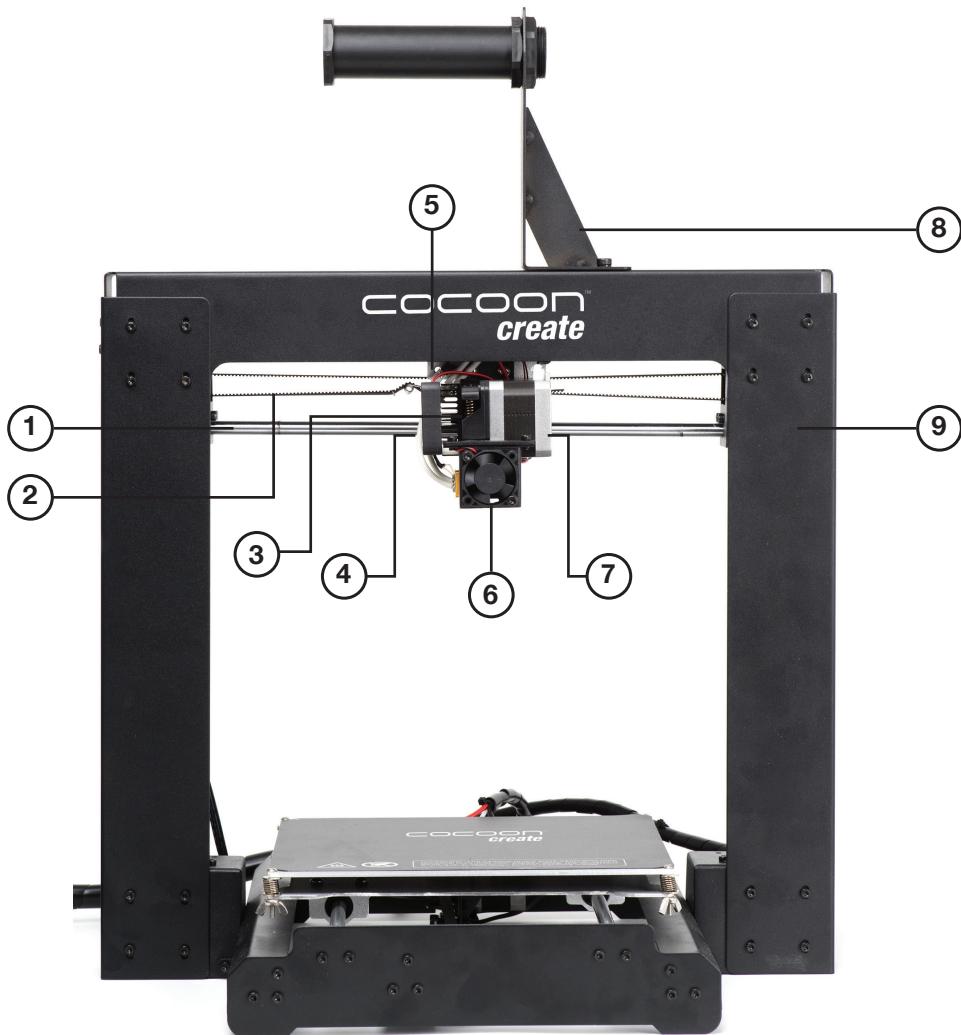
# Product Overview

**Printer Front View**



1. X Rail
2. X Belt Drive
3. Extruder Spring Lever
4. Extruder Cooling Fan

5. X Stop (Micro Switch)
6. Filament Cooling Fan
7. Extruder Driving Motor

8. Filament Spool Holder Assembly
9. Tower Frame
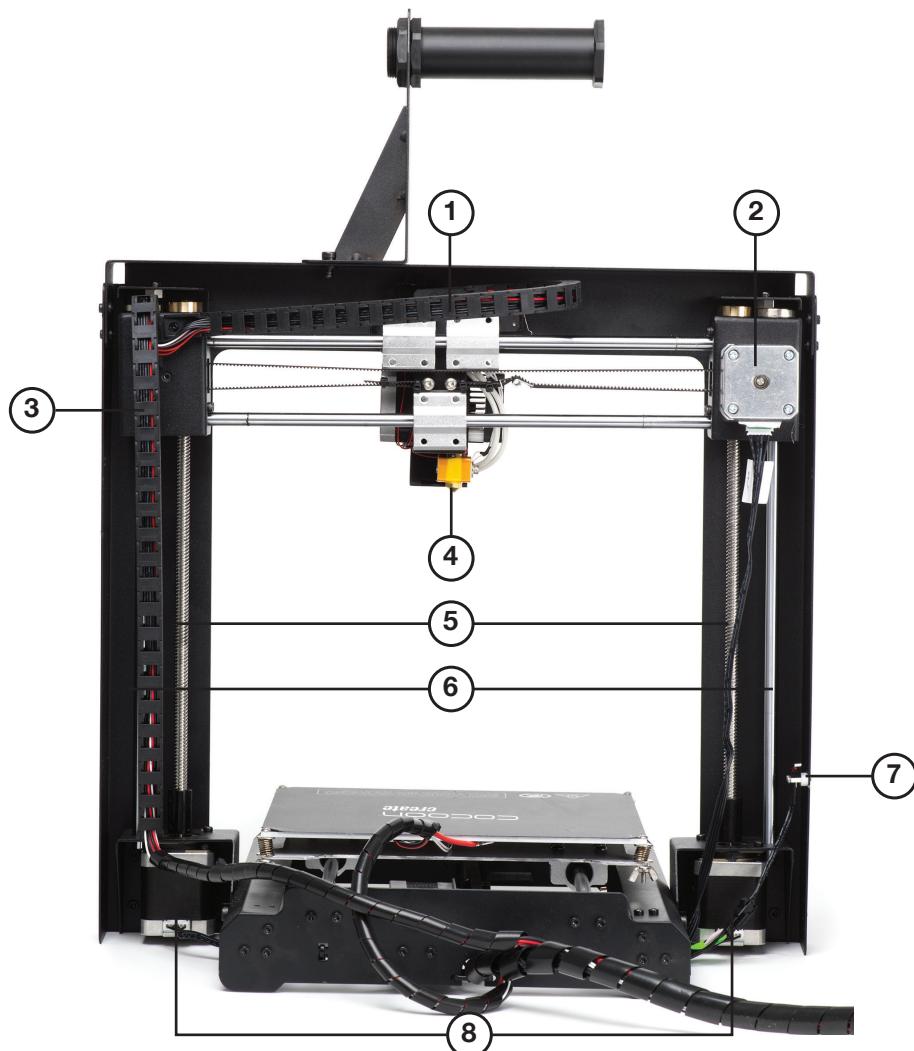
**After Sales Support**

# Product Overview

## Printer Rear View



1. X Cable Track
2. X Axis Stepper Motor
3. Z Cable Track
4. Extruder
5. Z Drive Screw
6. Z Rod
7. Z Stop (Micro Switch)
8. Z Axis Stepper Motor

# Product Overview

## Print Bed



1. Y Stop (Micro Switch)
2. Heated Print Bed
3. Printing Mat
4. Levelling Wing Nut Assembly
5. Print Bed Frame
6. Y Axis Stepper Motor
7. Y Belt Drive
8. Y Rail

# Product Overview

**Control Box**



| | | |
|---|---|---|
| **1.** LCD Display | **3.** Micro SD Port | **5.** Power Socket |
| **2.** Navigation Dial | **4.** Mini USB Port | **6.** Power Switch |

# Quick Guide

## Construct and Connect

1. Place the Control Box, Extruder Tower and Heated Print Bed on a clean, level work space. Carefully remove the cable tie that holds the Extruder in place



2. Insert the Heated Print Bed lengthways between the legs of the Extruder Tower and align the bolt holes on each side. Using the Hex Wrench, insert and tighten the two external bolts. Place printer on its side, then insert and tighten the two internal bolts



3. Carefully cut the cable tie on the left side of the Heated Print Bed Base. Then, on the left side, insert the long cable connector marked 'A' into the motor connector of the Extruder Tower marked 'A'



4. On the left side, insert the short cable connector marked 'B' into the motor connector of the Extruder Tower marked 'B'

5. On the right side, insert the white connector marked 'C' into the motor connector of the Extruder Tower marked 'C'

6. On the left side, insert the black connector marked 'D' into the black connector of the Extruder Tower marked 'D'

7. Remove the outer plastic nut from the Filament Spool Holder, then insert the threaded into the large hole of the Filament Spool Holder Stand. Tighten the plastic nuts on each side to secure

8. Position the Filament Spool Holder Stand on top of the Extruder Tower, then insert the bolts and tighten with the Hex Wrench

9. Ensure the power switch is in the OFF position, then insert the Power Supply Cable into the back of the Control Box. Insert the other end into a nearby mains power outlet. Turn on the power outlet, then turn on the 3D Printer, using the switch on the back of the Control Box

# Quick Guide

**10.** Wait for the printer to initialise, with **Printer Ready** shown at the bottom of the LCD. Press the Navigation Dial to enter the menu system. Rotate the Dial until **Position** is highlighted, then press the Dial to select. Rotate the Dial until **Z. Pos. Fast** is highlighted, then press to select. Rotate the Dial clockwise, raising the Extruder so that you can easily access the Heated Print Bed. Turn off your 3D Printer

## Levelling the Bed

**1.** Squeeze each corner of the printing bed, and tighten each wing nut to reduce the distance between plates to a minimum. Wipe the top of the Heated Print Bed with a clean, lint-free cloth to ensure no dust particles or other objects are on the Heated Print Bed. Remove all the white tape covering the X, Y, Z axis stop micro switch prior to the next step

**2.** Using the switch on the back of the Control Box, turn on the Cocoon Create 3D Printer. Once initialised, press the Navigation Dial to enter the menu system. Navigate to and select **Position** then select **Home All**.  Your 3D printer will now try to align to the home position, on the front left corner of the Heated Print Bed. Once aligned, turn off the 3D printer

*NOTE: Immediately turn off the machine if any of the motors are jamming or forcing against the chassis*

**3.** With the printer powered off, check the Z axis level. The X rail should be parallel to the Heated Print Bed. If it is not, manually rotate the right hand side Z axis stepper motor to level it

**4.** Manually move the Extruder to one corner of the Heated Print Bed to check the distance between the nozzle and Heated Print Bed. Rotate the wing nut to raise the Heated Print Bed. The distance between the Extruder and bed should be 0.1mm or roughly the thickness of an A4 sheet of paper.  The paper should have very slight friction as you move it under the Extruder but should still slide freely

**5.** Move the Extruder to each of the other corners and repeat the process. Move the Extruder to the centre of the Heated Print Bed and test the gap with a sheet of paper. Keep adjusting and testing until the gap is uniform on all points of the bed

*IMPORTANT NOTE:  Levelling the bed is an extremely important step and if it is not done correctly every time, your prints will fail or be of poor quality*

**After Sales Support**

**COCOON** *create*
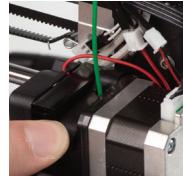
# Quick Guide

## Printing a File

1. Turn on the 3D Printer, using the switch on the back of the Control Box. Once initiated, press the Dial to enter the menu system. Navigate to, and select **Position** > **Z Pos. Fast.** Turn the Dial to the right to raise the Extruder to about 30mm. Press the Dial to return to the previous menu and select back

2. Navigate to, and select **Quick Settings**. Navigate to and select **Preheat PLA**. Wait for the 3D printer to automatically return to the home screen, after approximately 15-30 seconds

3. Check the temperature of the Extruder and plate on the LCD. When the temperatures are within a few degrees of the target, you are ready to insert the filament.

**NOTE:** *Preheat PLA will still be shown, even after temperatures have been reached. Insert the end of the filament into the hole on the Extruder as shown*



4. While holding the filament in place with one hand, press the Dial to enter the menu system. Navigate to and select **Extruder** > **Extr. Position**. Gently push down on the filament while turning the Dial clockwise. You will feel the filament being pulled into the Extruder. Continue until a few centimetres have extruded out of the nozzle. Carefully clean the extruded filament off the Heated Print Bed and nozzle using a cloth taking care not to burn yourself on the hot surfaces

5. Insert the included MicroSD card into the slot on the side of the Control Box. Press the Dial to enter the menu system. Navigate to and select **SD Card** > **Mount Card** > **Print File**

# Quick Guide

**6.** You can now select one of our pre-loaded files below by navigating to, and selecting the file name:

| | | | | |
|---|---|---|---|---|
| |  |  |  |  |
| **File Name:** | Unicorn | M3 Thumbscrew | Vase | CocoonBot |
| **Size (LxWxH):** | 47x19x51mm | 19x19x4mm | 50x50x100mm | 26x34x15mm |
| **Infill:** | 100% | 100% | 0% | 20% |
| **Resolution:** | 0.2mm | 0.1mm | 0.2mm | 0.1mm |
| **Print Time:** | 50min | 6min | 1hr 43min | 36min |

**7.** Your printer will now commence heating the Extruder and Heated Print Bed to the correct printing range, before returning to the home position and commencing the print

**8.** Once your printer has completed printing the file, the nozzle and Heated Print Bed will commence cooling down. Once cooled, turn off your 3D printer and carefully remove the 3D printed object using the supplied Scraper and gently lifting the object with your hand



**Always take care when using the Scraper. Never use excessive force or scrape towards your hand.**

*NOTE: Always keep the Scraper as level as possible. Scraper can damage the print mat if not used correctly*

# Introduction

## 1.1 What is 3D Printing?

3D Printing is a process whereby a real object is created from a 3D Design. Fused Filament Fabrication (FFF) is one of the most common technologies used by 3D Printers to achieve this. FFF works on an 'additive' principle by laying down material, such as PLA or ABS filament, in layers to create a 3D object.

## 1.2 From 3D Design to Printed 3D Object:
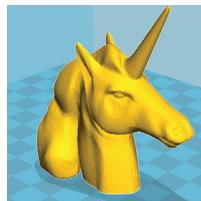
3D Design ➡ Slice to GCODE ➡ Setup 3D Printer ➡ 3D Printing Process ➡ 3D Printed Object



### 3D Design

Having a 3D Design is the first step to creating a 3D Object. There are a number of free 3D Designs that can be downloaded and printed for personal use. Websites such as Thingiverse.com, Pinshape.com and YouMagine.com are a good place to start, but there are many more available!

Alternatively, you can create your own 3D Design using 3D Modelling Software. Programs such as SketchUp or Blender allow this, however this can take some practice before successfully modelling your 3D Design. After you've finished your 3D Design, you can export it as an .STL file, to be imported into a 'Slicer' program such as Cura.



### Slice to GCODE

A 'Slicer' converts a 3D Design (usually in .STL or .AMF format) into individual layers. It then generates the machine code (such as GCODE) that the 3D Printer will use for printing.

We recommend Cura, as it uses a simple interface and includes a large amount of support material.
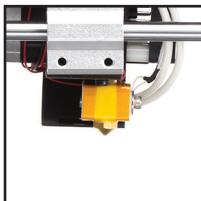
# Introduction

**Setup 3D Printer**

It's important to set up your printer before every print. We've included some steps on page 20 in section 1.3 'Good 3D Printing Practices' for you to quickly refer to each time.

**3D Printing Process**

Now that you've loaded a GCODE file onto the printer, it will commence heating up the Extruder and Heated Print Bed, then start printing. While printing, the nozzle will move along the X and Y axis as each layer is printed. This can take some time, depending on the size of the 3D model, print speed of the 3D printer and resolution of each layer.

Here is an example of how size, infill, and resolution can effect printing time:

| | | | | |
|---|---|---|---|---|
| Dimensions: | 62.5 x 25.75 x 67.75mm | | 70 x 70 x 140mm | |
| Infill: | 20% | **0%** | 20% | 20% |
| Resolution: | 2mm | 2mm | 2mm | **1mm** |
| Print Time: | 53min | 39min | 6hr 40min | 13hr 12min |
| Filament Used: | 4m | 2.54m | 45m | 45.67m |

**3D Printed Object**

Once the printer has completed all of the layers, you will have a 3D Printed Object. The results are certainly worth it, with 3D models ranging from simple to complex.

The Cocoon Create 3D Printer is compatible with a broad range of filament types and is capable of printing 3D models ranging from small and simple to complex objects with moving parts. What you print is limited only by your imagination.

# Introduction

## 1.3 Good 3D Printing Practices

Please refer to these steps during every print to ensure the best quality.

### Before Printing

1. Ensure your GCODE file is correctly setup, filament diameter is set to to 1.75mm and correct temperature settings for the material you are using (refer to table 3.1, page 27)
2. Check all cables and connectors are connected
3. Check that Heated Print Bed is level and approximately 0.1mm from the nozzle
4. Check that Heated Print Bed is clean and dust free and nozzle does not have excess material stuck to it
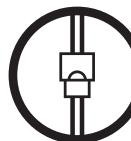
| Always check GCODE file settings match filament and printer | Always check Print Mat is clean and there is no residual plastic | Always check Extruder & Print Bed Temperatures are set for correct filament type | Always check all cords are connected | Always check Extruder distance is 0.1mm from Print Mat | Always check Print Bed is level |
|---|---|---|---|---|---|
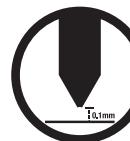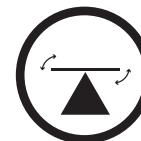
### During Printing

1. Closely monitor the first layer being printed – Most problems will occur in the first layer. If problems occur, stop the 3D Printer at the power outlet
2. Do not leave the 3D Printer unattended while in operation
3. Keep hands clear while machine is operating

### After Printing

1. Wait for the Heated Print Bed and Extruder to cool down
2. Carefully remove 3D Object. Never use Scraper towards your body. Keep Scraper evenly flat when scraping to avoid damage to the Heated Print Bed. Damaged Heated Print Bed will affect adhesion for future prints
3. Store 3D printer in a clean, dry environment

# Installation

## 1.4 Unboxing and Assembly

Your Cocoon Create 3D Printer is packed tightly and carefully -- take your time unpacking it.

To view a step by step video of the Unboxing and Assembly, visit

**cocooncreate.com.au/3dprinter/videos**



1.  Place the Control Box, Extruder Tower and Heated Print Bed on a clean, level work space. Carefully remove the cable tie that holds the Extruder in place



2.  Insert the Heated Print Bed lengthways between the legs of the Extruder Tower and align the bolt holes on each side. Using the Hex Wrench, insert and tighten the two external bolts. Place printer on its side, then insert and tighten the two internal bolts



3.  Carefully cut the cable tie on the left side of the Heated Print Bed. Then, on the left side, insert the short cable connector marked 'A' into the motor connector of the Extruder Tower marked 'A'



4.  On the left side, insert the long cable connector marked 'B' into the motor connector of the Extruder Tower marked 'B'



5.  On the left side, insert the black connector marked 'C' into the black connector of the Extruder Tower marked 'C'

# Installation



6. On the right side, insert the white connector marked 'D' into the motor connector of the Extruder Tower marked 'D'
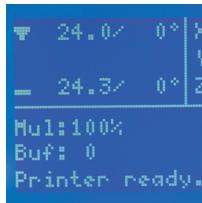


7. Remove the outer plastic nut from the Filament Spool Holder, then insert the threaded end into the large hole of the Filament Spool Holder Stand. Tighten the plastic nuts on each side to secure



8. Position the Filament Spool Holder Stand on top of the Extruder Tower, with the Filament Spool Holder centred on the stand. Insert two bolts and tighten with the hex wrench until secure



9. Ensure the power switch is in the OFF position, then insert the Power Supply Cable into the back of the Control Box. Insert the other end into a nearby mains power outlet. Turn on the power outlet, then turn on the 3D Printer, using the switch on the back of the Control Box
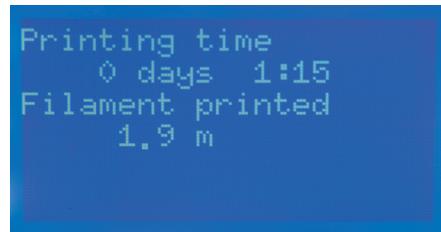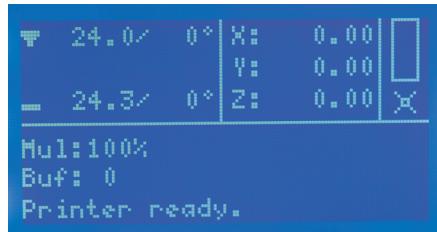


10. Wait for the printer to initialise, with 'Printer Ready' shown at the bottom of the LCD. You've successfully setup your Cocoon Create 3D Printer

# Installation

## 1.5 Control Panel

### 1.5.1 Main Display



**Extruder Temperature:** (Current °C / Set °C)

**Print Bed Temperature:** (Current °C / Set °C)

**Extruder Co-ordinates:** X, Y, and Z (mm)

**Mul:** Displays current speed multiplier. Default value is 100%

**Buf:** Displays current GCODE steps in the buffer

**Message Display:**

- **Printer Ready:** Printer is initiated and awaiting input
- **Idle:** Printer currently in idle state
- **Stepper Disabled:** Motors are disabled and free to be moved by hand
- **Preheat PLA / Preheat ABS:** Extruder and bed temperatures set and active
- **Printing…:** 3D Object is being printed, % displayed

**Printing Time:** Total time spent printing

**Filament Printed:** Total length of filament fed through the Extruder in metres

### 1.5.2 Menu Map

**Quick Settings:**

- **Home All:** Returns Extruder and bed to home position using stop switches
- **Speed Mul.:** Default 100%. Motor speed multiplier. Speed of the motor is multiplied by this rate
- **Flow Mul.:** Default 100%. Printer flow multiplier. Amount of filament extruded is multiplied by this rate
- **Preheat PLA:** Sets Extruder to 215°C and Heated Print Bed to 60°C
- **Preheat ABS:** Sets Extruder to 245°C and Heated Print Bed to 90°C
- **Cooldown:** Sets Extruder and Heated Print Bed to 0°C
- **Disable Stepper:** Disables all motors so they are free to be moved by hand

# Installation

**Print File:** If an SD Card is mounted, files can be quickly accessed here.

**Position:**

- **Home All:** Returns Extruder and bed to home position using stop switches
- **Home X / Y / Z:** Returns specific axis to home position using stop switches
- **X / Y / Z Pos. Fast:** Quickly adjust the location of a specific axis
- **X / Y / Z Position:** Precisely adjust the location of a specific axis
- **Extr. Position:** Adjust the Extruder driving motor to feed, insert or remove filament

**Extruder:**

- **Bed Temp:** Manually set the bed temperature 30°C and 120°C
- **Temp. 1:** Manually set the Extruder temperature between 170°C and 260°C
- **Extruder 1 Off:** Quickly return the Extruder temperature to 0°C
- **Extr. Position:** Adjust the Extruder driving motor to feed, insert or remove filament
- **Set Origin:** Returns Extruder position value to 0mm

**Fan Speed:** Adjust the Extruder cooling fan to your desired percentage

**SD Card:**

- **Print File:** Select a file to be printed from the MicroSD Card
- **Mount Card:** If inserting the MicroSD Card after the 3D printer has initiated, card can be mounted using this setting
- **Unmount Card:** Safely unmount the MicroSD Card
- **Delete File:** Select a file to be deleted from the MicroSD Card

**Debugging:**

- **Echo:** Off. Echo GCODE steps to confirm what has been received
- **Info:** On. Show printing information on the main display
- **Errors:** On. Show errors on the main display
- **Dry Run:** Off. Perform the GCODE steps without extruding any filament

# Installation

### 1.5.3 Default Configuration

The following settings show the default configuration values of the Cocoon Create 3D Printer.

We've set these to provide optimal quality for your 3D printed models, however, if you wish to adjust these settings, we recommend this to be done by only highly experienced users, as print quality and 3D printer durability may be affected.

- **General**
  Baudrate: 115200
  Stepper Inactive: 360s
  Max Inactive: 0s

- **Acceleration**
  Print X: 1000
  Print Y: 1000
  Print Z: 100
  Move: X
  Move: Y
  Move: Z
  Jerk: 20.0
  Z-Jerk: 0.3

- **Feedrate**
  Max X: 200
  Max Y: 200
  Max Z: 2
  Home X: 40
  Home Y: 40
  Home Z: 2

- **Extruder**
  Steps/mm: 96.0
  Start FR: 20
  Max FR: 50
  Accel: 5000
  Stab. Time: 1
  Wait Units: 0mm
  Wait Temp.: 150C
  Control: Dead Time
  DT/PID P: 7.00
  PID I: 2.00
  PID D: 40.00
  Drive Min 40
  Drive Max 230
  PID Max: 255

- **Store to EEPROM:** Save configuration settings to the internal storage, to be re-loaded on reboot

- **Load f. EEPROM:** Return configuration settings to the last reboot

# Setup

## 2. Levelling the Heated Print Bed

It is important to ensure the Heated Print Bed is level and set to approximately 0.1mm distance prior to printing.

1. With the printer initialised, press the Navigation Dial to enter the menu system

2. Rotate the Dial until **Position** is highlighted, then press the Dial to select

3. Rotate the Dial until **Z. Pos. Fast** is highlighted, then press to select

4. Rotate the Dial clockwise, raising the Extruder so that you can easily access the Heated Print Bed

5. Turn off your 3D Printer

6. Check the Z axis level. The X rail should be parallel to the Heated Print Bed. If it is not, manually rotate the right hand side Z axis stepper motor to level it

7. Squeeze each corner of the Heated Print Bed, and tighten each wing nut to reduce the distance between plates to a minimum

8. Wipe the top of the Heated Print Bed with a clean, lint-free cloth to ensure no dust particles or other objects are on the surface

9. Turn on the 3D Printer, using the switch on the back of the Control Box

10. With the printer initialised, press the Navigation Dial to enter the menu system

11. Navigate to and select **Position** > **Home All**. Your 3D printer will now try to align to the 'home' position, on the front left corner of the Heated Print Bed

12. Once aligned, turn off the 3D printer

13. With the printer powered off, move the Extruder to one corner of the Heated Print Bed to check the distance between these components

14. Rotate the wing nut to raise the Heated Print Bed. A good distance is 0.1mm, and will allow a sheet of paper to slide freely with slight friction between the Extruder and Heated Print Bed

15. Move the Extruder to each of the other corners and repeat the process. Move the Extruder to the centre of the Heated Print Bed and test the gap with a sheet of paper. Keep adjusting and testing until the proper gap is present at all points

*IMPORTANT NOTE: Levelling the bed is an extremely important step and if it is not done correctly every time, your prints will fail or be of poor quality*

# Setup

## 3. Filament

The Cocoon Create 3D Printer uses high quality PLA and ABS filaments.  Only use quality Cocoon Create filaments as using generic filaments may damage your 3D Printer, affect the print quality and may void your warranty.  Extra filament including multiple colours and types are available from the online store at **store.cocooncreate.com.au**.

For beginners, we recommend using PLA materials as they are easy to print and does not produce fumes when printing.  Before using other types of filaments, ensure you understand the advantages and disadvantages of other filaments and their proper usage.

## 3.1 Filament Types

| Filament Type | PLA | ABS | PVA | HIPS |
|---|---|---|---|---|
| Extruder Temp: | 215ºC | 240ºC | 175ºC | 240ºC |
| Heat Bed Temp | 60ºC | 90ºC | 50ºC | 60ºC |
| Official Name: | Polyactic Acid | Acrylonitrile Butadiene Styrene | Polyvinylv Alcohol | High Impact Polystyrene |
| Printing Level: | Basic | Advanced | Advanced | Advanced |
| Advantages: | • Bioplastic<br>• Non-Toxic<br>• Less warping issues<br>• Hard or soft/ flexible variants<br>• High print speed | • Smooth finish<br>• Durable<br>• Ideal for mechanical parts | • Water soluble<br>• Non-Toxic | • Lightweight<br>• Cost effective<br>• Dissolvable in Limonene |
| Disadvantages: | • Low heat resistance<br>• Easier to break than ABS<br>• Requires thicker walls than ABS | • Petroleum-based<br>• Non-biodegradable<br>• Heated Print Bed required<br>• Fumes when printing<br>• Sunlight deterioration | • Sensitive to high heat. **Do not heat over 190C** | • Only available in white |
| Finishing: | • Sanding possible<br>• Limited gluing | • Easy sanding<br>• Easily soluble in acetone | • Gluing Only | • Gluing Only |

# Setup

**Alternative types of PLA**

These PLA Alternatives require advanced knowledge of the 3D Printer. Please visit

**www.cocooncreate.com.au** for details of filament types and parameters.

*NOTE: Only use Cocoon Create Filament with your 3D Printer, using other filaments may damage your 3D Printer and may void your warranty.*
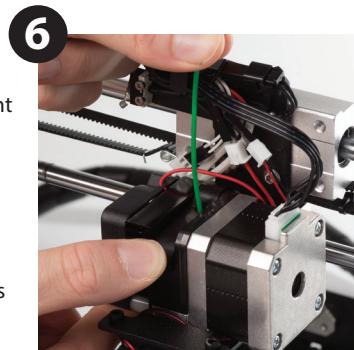
### 3.2 Inserting Filament:

1.  With the printer initialised, press the Navigation Dial to enter the menu system
2.  Navigate to, and select **Quick Settings**
3.  Navigate to, and select **Preheat PLA** or **Preheat ABS** - depending on the material in use
4.  Wait for the 3D printer to automatically return to the home screen, after approximately 15-30 seconds
5.  Check the temperature of the Extruder and plate on the LCD. When the temperatures are within a few degrees of the target, you are ready to insert the filament

*NOTE: Preheat PLA/ABS will still be shown, even after temperatures have been reached*



6.  Carefully insert the included filament sample into the hole on the Extruder as shown
7.  While holding the filament in place with one hand, press the Dial to enter the menu system
8.  Navigate to and select **Extruder** > **Extr. Position**.
9.  Gently push down on the filament while turning the Dial clockwise. You will feel the filament being pulled into the Extruder. Continue until a few centimetres have extruded out of the nozzle
10. Carefully clean the extruded filament off the Heated Print Bed and nozzle

*WARNING: Extruder and Heated Print Bed will still be hot*

# Setup

## 3.3 Removing Filament:

1. With the printer initialised, press the Navigation Dial to enter the menu system.

2. Navigate to, and select **Quick Settings** > **Preheat PLA** or **Preheat ABS** depending on the material.

3. Wait for the 3D printer to automatically return to the home screen, after approximately 15-30 seconds.

4. Check the temperature of the Extruder and plate on the LCD. When the temperatures are within a few degrees of the target, you are ready to remove the filament.

**NOTE:** *Preheat PLA/Preheat ABS will still be shown, even after temperatures have been reached.*

5. Depress the Extruder spring lever. Push down on the filament gently so that it starts to extrude from the nozzle.

6. When filament is extruding, quickly, but gently pull the filament upward. If it is done correctly, the end of the filament will pull out into a thin hair like strand.

**NOTE:** *The aim is to sever the filament where is it soft (inside the nozzle). Swift upward action will ensure that the filament does not cool down before exiting the nozzle inlet. If the filament gets stuck, please see the troubleshooting section.*

## 3.4 Storage:

Both PLA and ABS filaments are sensitive to moisture. If stored in a humid environment, the quality of the filament can deteriorate, with a tendency to bubble and spurt from the nozzle tip. This will affect print quality, so when storing PLA or ABS for an extended period of time, follow these tips.

- Store in a sealed, air tight bag/container. We recommend a vacuum seal bag
- Store with a silica gel (moisture absorber) pack. One is included with each filament spool

## 4. Using the Micro SD Card

## 4.1 Loading files from your Computer:

1. Insert your MicroSD Card into a card reader or memory card slot and connect to your computer (refer to your computer manual for further instructions if required)

2. The Micro SD Card will appear as a folder (refer to your computer manual for further instructions if required)

3. Copy your converted GCODE files to the Micro SD card, then safely remove

# Setup

## 4.2 Printing from the Micro SD Card:

1.  Insert the included MicroSD card into the slot on the side of the Control Box, ensuring that the metal contacts are facing upwards

2.  Press the Navigation Dial to enter the menu system

3.  Navigate to and select **SD Card** > **Mount Card** > **Print File**

4.  Navigate to your desired GCODE File, then press the Dial to commence printing

Your Cocoon Create 3D Printer will now heat up the Extruder and Heated Print Bed before printing your 3D object

**NOTE:** *If selecting 'Print File' shows no files, chances are the SD Card has not been mounted. Select 'Unmount SD Card', then 'Mount SD Card' to remount your SD Card into the side of your Control Box*

**NOTE:** *If Extruder temperature is set at 0°C after attempting to print a file, the imported GCODE may have some compatibility issues with the printer. Manually change the Extruder temperature using* **Extruder** > **Temp 1** *or* **Quick Settings** > **Preheat PLA** *or* **ABS** *depending on material used*

## 5. Removing 3D Objects from the Bed

When the Cocoon Create 3D Printer has completed printing your 3D printed object, it's time to remove the object from the bed.

1.  Wait until the Extruder and Heated Print Bed have cooled down to room temperature. This will allow the plastic to contract, making it easier to remove

2.  Using the Scraper, carefully 'wedge' under one of the edges of the 3D object. Take care to keep the Scraper flat with the bed, as to not cause damage to the Print Mat

3.  As one edge begins to lift, gently manoeuvre the Scraper forward to continue to 'wedge' between the 3D object and the Heated Print Bed

4.  Once the part has fully separated from the bed, it is safe to remove

**NOTE:** *Never scrape towards your body, or injury could occur*

**NOTE:** *Removing a 3D object, particularly one with a large surface size, can be a long process. Never rush when removing a 3D printed object*

**TIP:** If you are persistently having trouble removing 3D printed objects, try the following:

*   Use a layer of blue masking tape (designed to not leave behind residue) on the Heated Print Bed. Be sure to remove and replace after each print

*   Check if the printing mat is damaged and replace if needed

Replacement mats can be purchased at **store.cocooncreate.com.au**
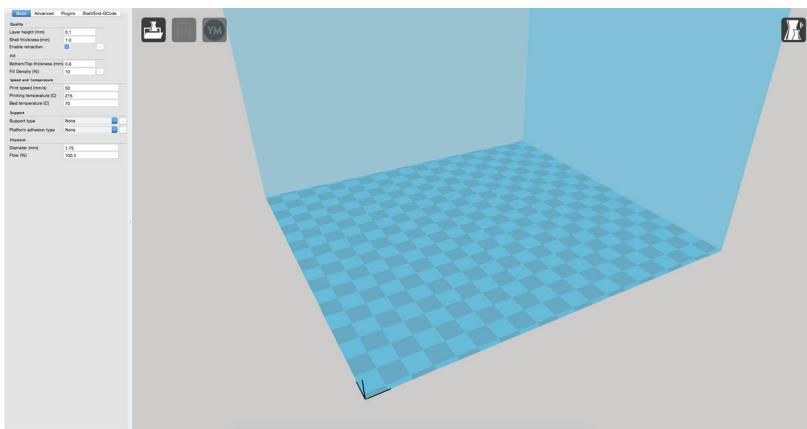
# Using Cura

## 6. Creating a GCODE File using Cura

A GCODE file provides the essential temperature settings and movement information for your Cocoon Create 3D Printer. A number of 'Slicer' software exists, which convert a 3D model file into a series of 3D printable layers. Cura is a free Open Source slicer software released on the AGPLv3 license. It's available on MacOS X and Windows.

### 6.1 Installing Cura

**1.** Install the file located on your included MicroSD card or download Cura from **cocooncreate.com.au/downloads**



**2.** Select a destination folder for the install, then press 'Next'

**3.** Select the components to install, then press 'Install'

**4.** Once complete, press 'Finish'

**5.** Refer to your computer manual for further instructions on the above steps if required
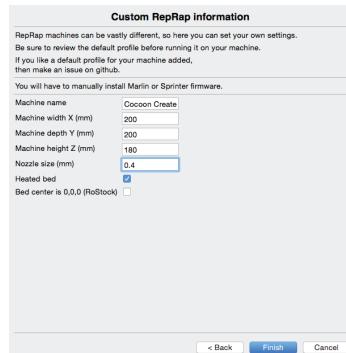
# Using Cura

**6.2 Adding your Cocoon Create 3D Printer to Cura**

1. On first launch, Cura will ask you to add a 3D printer. Alternatively, you can add a 3D printer by selecting **Machine** > **Machine Settings** > **Add New Machine**

2. Select **Other**

3. Select **Custom…**

4. Enter Machine name as 'Cocoon Create', or your desired name

5. Input Machine width and depth as 200

6. Input Machine height as 180

7. Change Nozzle size to 0.4mm

8. Ensure Heated Print Bed is ticked

9. Select 'Finish'

**6.3 Loading a 3D Design into Cura**

1. Clear the platform using **File** > **Clear platform…**

2. Then, load your 3D Design file onto the platform using **File** > **Load model file…**

3. Load any file with the following extensions: .STL, .OBJ, .DAE, .AMF

**TIP:** You can load more than one file onto the platform. Cura will automatically move the additional files to fit.

**NOTE:** *Files in grey are outside of the print area and need to be moved or scaled to fit.*

**6.4 Viewing and Positioning your 3D Model in Cura**

**Changing the View:**

- **Zooming In/Out:** Use the mouse scroll wheel
- **Rotating the View:** Hold down the right mouse button and move the mouse

**Move Position:**

Move your model to change where it is printed on the build plate. Do this by left clicking on the model and dragging it to the desired location. The black outlined corner represents the lower left hand corner of the Heated Print Bed on your printer.
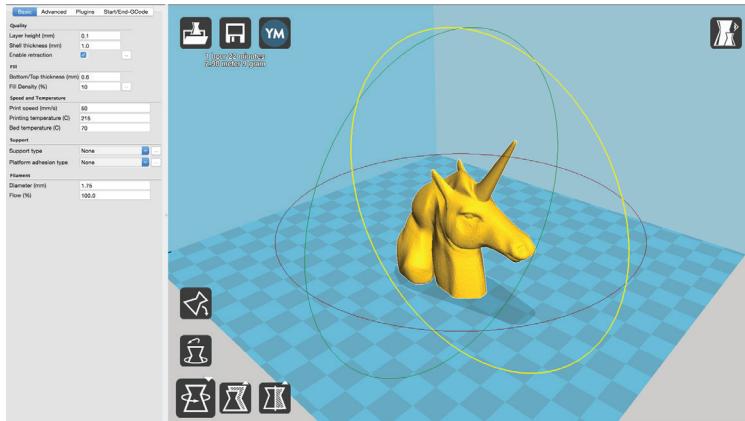
# Using Cura



### Rotate

The Rotate button will give you the ability to orient your model in along all three axes. Once you click the rotate button, three circles will surround your model. The red circle will allow you to rotate around the Z axis. The Yellow circle will rotate around the Y axis. The Green circle will rotate around the X axis.

### Lay Flat

The Lay Flat button will ensure that the flat portion of your print is securely attached to the bed. It is highly recommended to use this option after rotating your model in the Z direction, as it will help prevent adhesion issues during the print.

### Reset
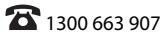
The Reset button will return your model to the original orientation as defined by the CAD program used to create the model.

### Scale

The Scale button displays the model dimensions, along with the ability to scale along the X Y or Z axes. Anything below the number 1.0 will reduce the objects size, while anything above the number 1.0 will increase the objects size. As a default, it will be set to uniform scaling. This will cause the X Y and Z axes to be scaled by the same amount when you make a change to any of them. To disable this, select the lock in the lower section of the scaling window.

**TIP:** Ensure your model is selected by left clicking on it. When selected, the model will have a white outline

### After Sales Support

![cocoon create logo]
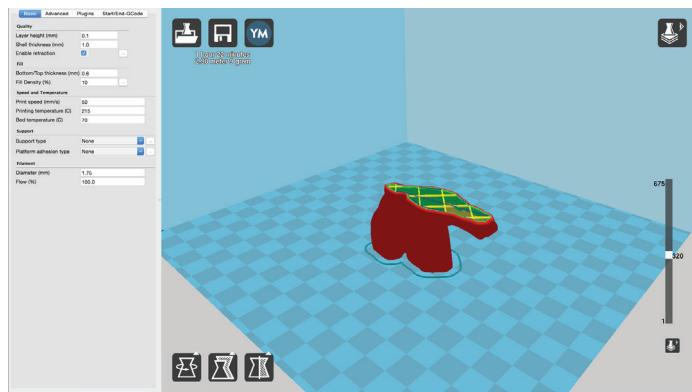
# Using Cura



### View Mode

This mode allows you to view your model in a variety of different ways.

This can be helpful for spotting issues before the print even starts.

### Normal

This is the standard view and shows the solid outer surfaces of the model.

### Overhang

Overhang mode shows where your model may need support material. Red highlighted areas show overhangs and more severe angles and areas where support material is recommended. Refer to page 38 for information on how to add support material to your model.

### Ghost

Ghost view mode makes the model translucent to allow you to see what is behind it.

### Xray

Xray is very similar to Ghost mode. It will allow you to see into objects, ensuring that inner details are correct.

### Layers

To view the toolpath of your print head and to ensure no skipped layers or gaps, use this option. Use the slide bar on the right hand side of the window to move up and down through the toolpath layers.

### After Sales Support
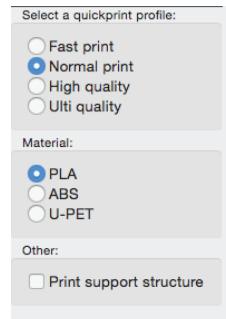
# Using Cura

## 6.5 Quick Print Settings

Quick Print mode allows you to quickly set the quality, material, diameter and support structure of your intended GCODE file.

Enter quick print mode by selecting **Expert** > **Switch to Quickprint…**

### High Quality

Designed to give greater detail and finer objects. This will have a smaller layer height, which will make each layer thinner, so that curves seem more natural and walls seem less noticeable.

### Medium Quality

Designed to give a medium resolution, by increasing the layer height and print speeds. This will make the organic curves slightly more step-like than the fine setting, but will reduce printing time.

### Fast Quality

Designed for the fast prints, where overall model finish is not of concern. Most commonly used for quick iteration of designs found in rapid prototyping.

### Material Selection

Choose your desired filament. The Cocoon Create 3D Printer includes a 10 metre sample of PLA to be used in your first print.
Other filaments are available for purchase at **store.cocooncreate.com.au**

### Diameter

Ensure this is always set to match the filament diameter. Your 3D Printer only accepts Cocoon Create 1.75mm filaments.
***NOTE:*** *Advanced users can fine tune the diameter value to allow for tolerance if needed*

### Printing Support Material

Your Cocoon Create 3D printer is able to print models that have angles and overhangs, some of these models will require support material depending on the overhang distance and angle. Turn this option on if your model could benefit from support material.

![COCOON create logo]

# Using Cura

### 6.5.1 Exporting your GCODE File:

Take note of the time and length required to print your 3D object. From the top menu, select **File** > **Save GCODE**. You can now save your GCODE to your desired directory.

*NOTE: Always reopen the GCODE file again in CURA to confirm that all slices of the model have been included. This will open in the 'layers' view, if you want to see your object, select 'normal' view*

### 6.6 Advanced Print Settings

Advanced print mode should not be used until you have gained sufficient experience with 3D printing. In order to have more control of your slicing and GCODE generation, switch to advanced print mode.

Enter advanced print mode by selecting **Expert** > **Switch to full settings…**

### Loading a Profile

When you first switch to Full Settings, Cura will revert to very generic settings. We recommend using our tested profiles that are available at **cocooncreate.com.au**. You will want to choose the profile that matches your filament and quality needs. Once downloaded, you can load the file into Cura by selecting File > Open Profile. Choose your desired profile. This will automatically update all of your settings for use with your printer.

### 6.6.1 Basic Tab Options
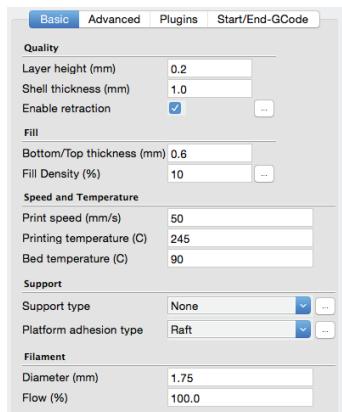
**QUALITY**

**Layer Height (mm)**

The thickness of each printed layer is known as the Layer Height. The smaller the layer height, the smoother curves will appear. Larger layer heights are better for bridging and overhangs. Smaller layer heights will also increase print time, as it will take more layers to complete the object.

**Shell Thickness (mm)**

This defines the number of vertical walls that comprise the outside of your model. We recommend keeping this set to multiples of your nozzle width. Your Cocoon Create 3D Printer is equipped with a 0.4 mm nozzle.

| Basic | Advanced | Plugins | Start/End-GCode |
|---|---|---|---|
| **Quality** | | | |
| Layer height (mm) | 0.2 | | |
| Shell thickness (mm) | 1.0 | | |
| Enable retraction | ☑ | … | |
| **Fill** | | | |
| Bottom/Top thickness (mm) | 0.6 | | |
| Fill Density (%) | 10 | … | |
| **Speed and Temperature** | | | |
| Print speed (mm/s) | 50 | | |
| Printing temperature (C) | 245 | | |
| Bed temperature (C) | 90 | | |
| **Support** | | | |
| Support type | None | … | |
| Platform adhesion type | Raft | … | |
| **Filament** | | | |
| Diameter (mm) | 1.75 | | |
| Flow (%) | 100.0 | | |

# Using Cura

**Enable Retraction**

Retraction tells your printer to pull filament out of the Extruder upon travel moves. Travel moves are when your print head moves from one area of the print, to another without laying down filament. We recommend keeping this on for all filament types, and adjusting the retraction length and speed for the specific filament.

**FILL**

**Bottom/Top Thickness (mm)**

Also known as Surface Layers; this will determine how thick the top and bottom layers are. A larger number here will create a thicker top and bottom which can be helpful for strength, bridging, and quality purposes. We recommend using a  number which is a multiple of your layer height.

*Eg: If using a layer height of 0.2mm, set the thickness to 0.4, 0.6 or 0.8mm*

**Fill Density (%)**

This number is expressed as a percentage. 0% will give a completely hollow print, while 100% will give you a completely solid object. We recommend 20% to 40% fill density as it is functional for most prints.

**SPEED AND TEMPERATURE**
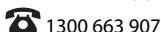
**Print Speed (mm/s)**

Your overall printing speed can be adjusted here. If no other speeds are determined in the later sections your printer will automatically default to this speed. This speed will be different, depending on what type of filament you are using. We recommend setting speed to 50mm/s.

**Printing Bed Temperature (C)**

When using different filament materials, you'll need to update the desired Extruder and Heated Print Bed temperature. Any temperatures specified here will be used to automatically set both the Extruder and Heated Print Bed. Your print will not begin until these temperatures are met. As a guide, refer to Table 3.1 on page 27 for temperature settings.

# Using Cura

**SUPPORT**

**Support Type**

Some models will require support material in order to print properly. This will usually occur when an object has an angle in relation to the Heated Print Bede between 0 to 45 degrees. It is highly recommended to orient your object so that it minimizes or eliminates the need for support.

### Touching Buildplate

This causes the support material to build up between the Heated Print Bed and the object. The red example is 'Touching Buildplate'

### Everywhere

This prints support material between the Heated Print Bed and object as well as between the object and itself. The green example is 'Support Everywhere'.



**Platform Adhesion Type**

Some models have a small surface area contacting the plate. This can create adhesion issues causing your part to pop off at some point during the print. To fix this, use either 'Brim' or 'Raft'. Raft is better used when a model has small contact points with the Heated Print Bed and overhangs.

### Brim

Brim will create a single layer of filament, contacting and surrounding your model. This will increase the surface area of the part contacting the build platform thereby preventing it from popping off the Heated Print Bed. Brim will also help in situations where you are seeing corner lift.
Brim settings can be adjusted by selecting '**...**' next to the tab.

### Raft

Raft will generate a layer of material underneath your object. Raft was more often used before the addition of heated plates to increase surface area. Raft settings can be adjusted by selecting '**...**' next to the tab.

# Using Cura

**FILAMENT**

**Diameter (mm)**

The filament diameter setting is one of the more important settings. Make sure that you update this value periodically with your average filament diameter. While your filament may be referred to as 1.75mm, it is more likely going to be near 1.7mm +/- 0.1mm. You will want this to be an accurate average, as it will allow your printer to correctly calculate how much filament it is pulling into the Extruder.

**Flow (%)**

This controls how much filament your printer is extruding in relation to speed. This setting is mainly used to adjust for filament density variations. Leave this value at 100% as changing it can lead to surface quality issues.

### 6.6.2 Advanced Tab Options

**MACHINE**
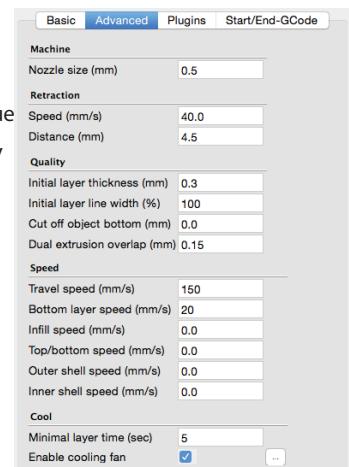
**Nozzle Size (mm)**

This defines your nozzle size. The slicing engine uses this value combined with your other settings to determine how quickly to feed filament into your Extruder. The Cocoon Create 3D Printer uses a 0.4mm nozzle.

**RETRACTION**

**Speed (mm/s)**

Retraction Speed determines the speed at which your filament is reversed out of the Extruder for travel moves and when changing direction during printing. We recommend keeping this set to 40mm/s.

| | | Basic | Advanced | Plugins | Start/End-GCode |
|---|---|---|---|---|---|
| **Machine** | | | | | |
| Nozzle size (mm) | | 0.5 | | | |
| **Retraction** | | | | | |
| Speed (mm/s) | | 40.0 | | | |
| Distance (mm) | | 4.5 | | | |
| **Quality** | | | | | |
| Initial layer thickness (mm) | | 0.3 | | | |
| Initial layer line width (%) | | 100 | | | |
| Cut off object bottom (mm) | | 0.0 | | | |
| Dual extrusion overlap (mm) | | 0.15 | | | |
| **Speed** | | | | | |
| Travel speed (mm/s) | | 150 | | | |
| Bottom layer speed (mm/s) | | 20 | | | |
| Infill speed (mm/s) | | 0.0 | | | |
| Top/bottom speed (mm/s) | | 0.0 | | | |
| Outer shell speed (mm/s) | | 0.0 | | | |
| Inner shell speed (mm/s) | | 0.0 | | | |
| **Cool** | | | | | |
| Minimal layer time (sec) | | 5 | | | |
| Enable cooling fan | | ☑ | | ... | |

**Distance (mm)**

Retraction Distance determines how much filament is pulled out of your Extruder on travel moves and when changing direction. You will want to adjust this depending on temperature settings and filament type. Higher thermal retaining filaments such as PLA behave better with a longer retraction distance. Anywhere from 1mm to 3mm is a good starting range.

# Using Cura

**QUALITY**

**Initial Layer Thickness (mm)**

This will control how thick your first printed layer height is printed onto the Heated Print Bed. Having a larger initial layer height will help prevent your part from popping off the plate. Your Cocoon Create 3D Printer auto levelling system could be affected if you change this from the standard profiles. ***Adjust at your own risk.***

**Initial Layer Line Width (%)**

This will control how wide your first extruded filament path is for the initial layer. A wider line width will help with bed adhesion. We recommend 125% as a good starting place. For models with moving printed in place parts, a smaller initial layer line width is recommended.

**Dual Extrusion Overlap (mm)**

This will determine how far your Dual Extruders will overlap when laying down material. This will help adhesion between the two different colours or types of filament. This setting is not applicable to the Cocoon Create 3D Printer, it is only for printers with dual Extruders.

**SPEED**

**Travel Speed (mm/s)**

This setting will determine how fast your print head moves while not extruding filament. A normal travel speed of 125 - 150mm/s is recommended.
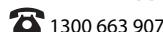
**Bottom Layer Speed (mm/s)**

This will control your initial layer speed. In general, a slower initial layer speed will help with first layer adhesion.

**Infill Speed (mm/s)**

This is how fast your print head speed will be while laying down the interior portion of your model. Faster speeds are usually tolerable here, as none of the infill will be visible from the outside of your object. However, if you go too fast compared to your inner and outer shells, you can have adhesion issues or globs of filament left behind from the print head.

# Using Cura

**Outer Shell Speed (mm/s)**

This will be the outermost surface of the model. This is the most important setting, as it controls the speed of your print head on the visible layers. As a general rule of thumb, the slower you go the better looking print you will get.

**Inner Shell Speed (mm/s)**

This affects vertical walls that are in between the outer shell and infill. This will not be visible but will help support the outer shell and the infill. We recommend keeping this speed setting between your infill speed and your outer shell speed.

**COOL**
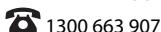
**Minimal Layer Time**

This will determine a minimum amount of time your printer will spend laying down each layer. If your layer print time falls below this your printer will automatically slow down to reach this time before moving onto the next layer. Tweaking this can help get cleaner, crisper prints.

**Enable Cooling Fan**

Enables operation of your Extruder's active cooling fan. The fan settings can be adjusted by selecting '**...**' next to the tab..

![cocoon create logo]
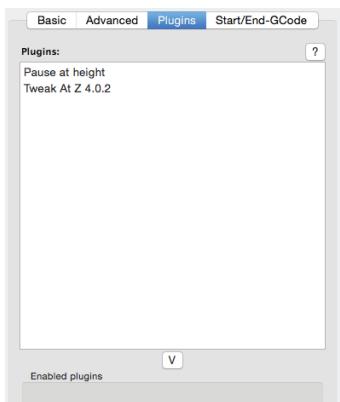
# Using Cura

### 6.6.3 Plugins Tab

Plugins are custom settings which will alter your print at specific points. The two that come pre-loaded with Cura are Tweak at Z, and Pause at Height. More plugins and information can be found here:

**http://wiki.ultimaker.com/Category:CuraPlugin**

To activate one of these highlight the desired plugin and click the drop-down arrow directly below the Plugins box.

**Tweak at Z**

Make basic changes at specified Z heights. You can determine the Z height or layer count at which you want to make a change. Then choose how you would like to change your settings. You can alter temperatures, fan speeds, and print speeds. Fine tuning these for specific STL files, can produce cleaner prints.

**Pause at Z Height**

Pause your print at a specified height. You can also specify where to move the print head and how much filament to retract. This will prevent 'blobs' from accumulating on your print while paused. This setting is most commonly used when switching colours of filaments in the middle of a print.

### 6.6.4 Start/End GCODE Tab

Custom Gcode allows for complex automatic printer movements and operations. By adding custom Gcode into the start or end of your file, you can alter how it prints. A comprehensive list of Gcode commands can be found here:

**http://reprap.org/wiki/G-code**

We recommend new users to leave this as provided.

# Using Cura

## 6.6.5 Expert Settings

Expert settings will give you more specific options for your retraction, skirt, active cooling, infill, support, brim, raft, and special settings. To gain access to this section you go to **Expert** > **Open Expert Settings** or on your keyboard press **Control + E**.

**Retraction**

Retraction pulls filament out of your nozzle when it is not extruding to prevent your print head from dripping on your object. This section is where you will control how your Extruder retracts its filament.

>**Minimum Travel (mm)**
>
>This sets the minimum travel distance of your print head in order to retract. If your print head is not moving this far during travel moves, it will not retract.
>
>**Enable Combing**
>
>This option prevents your print head from travelling over holes in the X/Y plane when printing.
>
>This will slightly increase print time, but will prevent strings from getting caught on the holes during travel moves. We recommend keeping this setting on.
>
>**Minimal Extrusion before Retracting (mm)**
>
>This will control the distance at which retraction occurs if the printing movement exceeds the minimum extrusion amount. This will prevent a retraction move, if your Extruder has not put out Xmm of filament since its last retraction.
>
>**Z Hop When Retracting (mm)**
>
>This will raise your print head Xmm while retracting. This setting helps prevent ooze, and strings from being deposited on your print.

# Using Cura

**Skirt**

Skirt creates a line around the outside of your object. This is most commonly used to prime the Extruder. In order to prevent missed filament at the beginning of a print, we recommend you leave this setting on.

### Line Count

This will define the number of loops the Skirt creates around the outside of your object. Smaller models will require more loops to properly prime the Extruder.

### Start Distance (mm)

This will define the distance away from your model that the skirt will be created. If using as an envelope to prevent drafts, it is recommended to be closer to your object.

### Minimal Length (mm)

This will define the minimum extruded line length for the skirt. This will over ride your line count, producing as many lines as required to reach the minimal length.

**Cool**

This section will define how your Extruder cooling fan will operate during the print. Your fan will not start until it has reached 25% or higher for speed settings. If your print speeds are slowed down due to minimal layer time, the fan will run between minimum and maximum speed based upon how much the layer is slowed down.

### Fan on at Full Height (mm)

The fan will not turn on for the first layer. The fan will then be scaled between 0% and your minimum fan speed based on layer height. It is especially helpful with high temperature retaining filaments such as PLA.
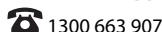
### Fan Speed Min (%)

Once the Z height is reached for Fan on at Full Height setting, this will be the speed your fan runs at.

### Fan Speed Max (%)

This is the fastest speed at which your fan will ever run. When your print speed is slowed down due to Minimal Layer Time settings, your fan will run between minimum and maximum speed. The maximum fan speed is reached when your printer speed is slowed by 50% or greater.

# Using Cura

**Support**

You define how your support material is generated here. You must have a support type in the 'Basic Settings; in order for these support settings to have an effect.

### Structure Type

You can choose between a 'Grid' or a 'Line' pattern for your support material. The grid will be a checkerboard pattern in the X and Y direction. The line option will produce lines along the Y-axis for support. The grid will provide stronger support than the line option, but will be harder to remove.

### Overhang Angle for Support (deg)

This will determine where support material is generated. In general you will be able to print a model with 45 to 90 degree angles in relation to the Heated Print Bed without support. We recommend leaving this setting at 45 degrees.

### Fill Amount (%)

This will determine how dense your support material is printed, similar to Infill Percentage. The higher percentage the better support, but it will be harder to remove the support material and will use more filament.

### Distance X/Y(mm)

This will determine how far away from your object in the X/Y plane that the support material is being placed.

### Distance Z (mm)

This will determine how far away your support material is from your object in the vertical direction. A smaller number here is better for support, but makes it harder to remove.

**Black Magic**

This section allows you to transform your model into a hollow shell, a single layer thick.
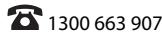
### Spiralize the Outer Contour

This causes your Z-axis to be constantly moving upward as you are printing your single outer wall shell. This results in no layer change lines, giving a much smoother surface. This setting is typically only used for artistic objects as they will be fragile.

### Only Follow Mesh Surface

This will cause your print to follow the outside of your model, building it completely hollow with a single wall outer shell. The only difference between this and Spiralize, is that the Z-axis moves regularly. That is, it prints a layer and then moves up to the next one.

**After Sales Support**

☎ 1300 663 907     💻 customercare@winplus.com.au     **Model Number** HE150272

# Using Cura

## Brim

Brim circles the base of the print while making contact, helping adhere the print to the Heated Print Bed. This is only one layer thick, and can be easily removed post-print. This section defines how the brim is formed when brim is activated in 'Basic Settings'.

### Brim Line Amount

This will determine the distance the brim will cover around the outside of your object. The more brim used, the better your part will adhere to the plate.

## Raft

Raft is a platform built underneath your object, designed to help adhesion and prevent warping. It will lay down support material, and then a platform on top of the supports. Your model will be built on top of this platform. The bottom surface of your printed object will not be as clean or as even when using this option. Raft is typically not recommended.

### Extra Margin (mm)

This determines the distance around the outside of your object that the raft is created. This can be helpful for ensuring no warping occurs in the lower layers.

### Line Spacing (mm)

This will determine the spacing between support lines for the raft. A small spacing makes the support structures closer together improving strength of the raft, but uses more filament.

### Base Thickness (mm)

This defines how thick your raft will be.

### Base Line Width (mm)

This will define how wide your support material is for the raft. This setting will determine how well the surface layers of the raft print.

### Interface Thickness (mm)

This will determine how thick the surface layers of the raft are. The surface layers are the platform that is built upon the supports.

### Interface Line Width (mm)

This will determine how wide the top layers of the platform will be. In general, you can keep this set to your nozzle size, as surface quality of the removable raft is not important.

# Using Cura

### Airgap

This will define the distance between your raft and your print. A larger gap will make your part easier to remove, but will make the bottom of your print look worse.

### Surface Layers

This will determine the number of layers that create the 'platform' of your raft. If you have a wide line spacing, you may want to increase this number to ensure a solid platform.

## Fix Horrible

These are some of the more advanced and experimental options. They are designed to help repair models with errors to make them suitable for 3D printing. However they may not always work. Please be cautious when using these options as they can have unintended effects on your print quality.

### Combine Everything (Type-A)

This will attempt to fix all external mesh errors, while keeping internal holes intact. This can accidentally fill in intentional internal holes.

### Combine Everything (Type-B)

This will ignore all internal holes of the model and only focus on the external holes. This is helpful when only the outside finish of the model is important.

### Keep Open Faces

This will ignore all manifold errors in the object. It can create issues generating the Gcode as Cura does not know how to interpret the open holes. This option should only be used if you are sure that the holes in the mesh are intended. In general, you should not use this option.

### Extensive Stitching

This causes Cura to automatically add triangle mesh in an attempt to fix manifold errors. This algorithm will greatly increase Gcode generation time and may end up adding in un-intended mesh. It is recommended that you repair your model through your CAD program before attempting this option.

# Servicing

## 7. Servicing your 3D Printer

### 7.1 Replacing the Print Mat

The Cocoon Create 3D Printer includes 2 x Print Mats. Additional Print Mats are available for purchase from **store.cocooncreate.com.au**

1.   Remove the current Print Mat by slowly peeling from one corner
2.   Using rubbing alcohol and a microfiber towel, carefully remove any residual adhesive left on the Heated Print Bed
3.   Peel the 3M backing from the new Print Mat
4.   Gently place the new Print Mat onto the Heated Print Bed, making sure to align with each edge. It is recommended to start at a corner to ensure no bubbles become present
5.   Use the Scraper to apply pressure on the new Print Mat, ensuring strong adhesion. Keep the Scraper level as it can damage the Print Mat
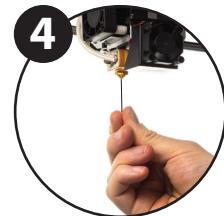
**TIP:** Warming the bed slightly may make removal easier. You can also use masking tape in lieu of a Print Mat. Blue painters tape is recommended.

### 7.2 Nozzle Cleaning

If your Extruder nozzle becomes blocked, follow these steps to clean the nozzle.

1.   Remove the existing filament using steps from section 3.3 'Removing Filament'
2.   Raise the Extruder using **Position** > **Z Pos. Fast**, then rotating the Dial clockwise
3.   Heat the Extruder to 215°C
4.   Using a clean 0.4mm drill bit, insert the drill bit into the nozzle by hand, twisting while inserting
5.   Remove the drill bit from the nozzle, twisting while removing
6.   Repeat steps '4' and '5' three times, then follow the steps in section 3.2 'Inserting Filament' to re-insert the filament into the nozzle

# Servicing

## 7.3 Routine Component Check

**Motor Connectors**

Loose motor connections could cause a 3D print to fail. It's a good habit to routinely check for any damage to the connectors or the black cables.

### Axis Lubrication

Regular lubrication of the X and Y Axis rods is important for the longevity and smooth running of your printer. The rods can be lubricated with oil or grease. We recommend using bearing grease, and to apply generously to both the X Axis and Y Axis rods.
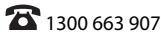
### Nuts and Bolts

Loose nuts and bolts could affect the geometry of the 3D Printer. Check that all nuts and bolts remain firmly tightened. If required, re-tighten using the hex wrench.

## 7.4 Storing

Always store your 3D Printer in a cool, dry location that is free from dust.

# Finishing

## 8. Finishing Techniques

### 8.1 Useful Tools/Equipment
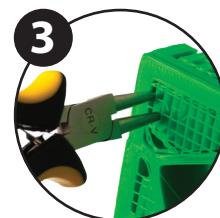
- Safety Goggles
- Mask
- Long-nose Pliers
- Wire cutters
- Sandpaper (100,200,300,400,500,600 Grit)
- Medium, fine and extra fine sanding sponges
- Acrylic based primer spray paint
- Acrylic based spray paint (colour of your choice)
- Acrylic based varnish spray paint

**NOTE:** *Different paint types are also compatible with PLA and ABS. Consult your local paint supplier for the best compatibility for your intended purpose.*

### 8.2 Removing Support Material

Even after support material is removed by hand, residual material will remain.

1. Ensure that safety goggles and a mask are worn for protection
2. Remove most of the support material by hand, taking care not to damage the 3D object
3. Using the long-nose pliers, carefully grasp some of the remaining material, then twist to remove. Repeat until no more material can be removed.
4. Using the wire cutters, cut away any excess filament.

# Finishing

## 8.3 Sanding

Though Fused Filament Fabrication (FFF) is capable of producing high quality objects straight out of the machine, the layer lines will be visible. Sanding will reduce these 'step' marks in the model.

1.   Ensure that safety goggles and a mask are worn for protection
2.   Starting with 100 grit sandpaper, carefully sand your 3D object using small, circular motions

**NOTE:** *PLA is less resistant to high temperatures caused by sanding. Take your time sanding your 3D object to avoid damage.*

3.   Using the medium sanding sponge, sand difficult to reach areas of your 3D object
4.   Gradually increase the sandpaper grid until the 3D print is smooth

**TIP:** *Spraying a primer filler will also help to reduce the layer lines on the 3D Object. This can then be sanded away and repeated, until the desired smoothness is achieved*

## 8.5 Surface Coating

Finally, to make your models more presentable, follow these steps;

1.   Ensure that safety goggles and a mask are worn for protection
2.   Ensure that your 3D object is clean and free from dust
3.   Spray your model using an acrylic based primer, following the directions of your primer
4.   Allow appropriate time to dry before applying additional coats
5.   Spray your model using an acrylic based paint of your colour. Allow appropriate time to dry
6.   Spray your model using an acrylic based varnish, following the directions of your varnish. Allow appropriate time to dry before applying additional coats.

**NOTE:** *Different paint types are also compatible with PLA and ABS. Consult your local paint supplier for the best compatibility for your intended purpose.*

Congratulations! Your 3D object now looks like a professionally finished product!

COCOON *create*

# Troubleshooting

## 9. Troubleshooting

If having any troubles when using the 3D printer, please refer to the troubleshooting guide below, or call customer support on **1300 663 907**.

You can also visit Please visit **cocooncreate.com.au/3DPrinter/Videos** for support videos and advanced tutorials.

### 9.1 Printing Problems

**Clicking sound from one of the stepper motors**

One of your stepper motors may not be connected properly. Check each connection and then inspect the cable routing for any faults

**File not printing**

Remove the Micro SD Card and insert into your Computer. Open the GCODE file using a text editor (eg. Notepad), and inspect if GCODE is readable or not. If file consists of multiple 'ÿÿÿÿ' symbol, then file has been corrupted. Try reformatting your Micro SD Card and reloading the GCODE file.

**NOTE:** *Reformatting will delete all files on your MicroSD card*

**Not extruding when starting to print**

Ensure that the nozzle temperature has been set to match your material
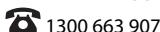
**Not sticking to the bed**

- Ensure that the bed is clean and free from dust
- Check that the bed is level and set to a distance of approximately 0.1mm
- If there are defects / deep scratches in the printing mat, it may need to be changed. Change the mat or use blue painters masking tape
- Add a brim or raft to your file
- Check that the bed temperature matches your material
- Ensure nozzle to bed gap is not bigger than 0.1mm. Material will solidify if gap is large.

**Models Warping/Curling**

- Check that the bed temperature matches your material
- Check the infill % of your GCODE. The higher the infill, the more likely to warp
- Add a brim or raft to your file

**After Sales Support**

☎ 1300 663 907     💻 customercare@winplus.com.au     **Model Number** HE150272

# Troubleshooting

**Model appears very 'stringy'**

- Ensure that the filament diameter is set to match your filament spool (1.75mm), then try creating a new GCODE before printing again
- Check you are using the correct temperature for the filament used

**Print stopped halfway**

- Check that the GCODE file is complete and not corrupt
- Check the original model file

**The motor does not stop at the end of axis**

Check that the stop micro switches are aligned with the motor mount and registering.

**The print head/bed does not move**

- Check the part can be moved by hand when stepper motor is disabled. Clear anything that is blocking the path
- Check that the motor is turning (using position menu). If motor turns but belt does not, you will need to tighten the gear motor gear nut. Please visit **cocooncreate.com.au/3dprinter/videos**

## 10.2 Connectivity Problems

**My 3D Printer isn't turning on**

Check that you have correctly inserted the power cable into the rear of the Control Box, the power cord is plugged into the mains outlet and that the Control Box power switch is on.

**Micro SD Card not shown**

Navigate to SD Card, select Unmount SD Card, then select Mount SD Card. Your Micro SD card should now appear in the menu.

# COCOON create

# Troubleshooting

## 10.3 Other Problems

**Filament stuck when removing**

Please visit **cocooncreate.com.au/3dprinter/videos** for a video on how to clear the nozzle with stuck filament.

**Where can I buy more filament?**

If you require more filament, visit our online store, **store.cocooncreate.com.au** to purchase various colours and types.

**What brand of filament should I buy?**

We recommend using only Cocoon Create branded filament.  Lower quality generic filaments can damage your 3D Printer and potentially void your warranty.

**What types of filament can my 3D Printer Print?**

The Cocoon Create 3D Printer accepts various types of filaments including PLA, ABS, PVA, HIPS and specialty filaments such as wood, flexible, luminous and conductive.  For our full range of filaments visit **store.cocooncreate.com.au**

## 10. Useful Terms

**ABS Filament**

Commonly used plastic filament with good strength and industrial properties

**Display**

Provides the main interface for operating the 3D Printer

**Firmware**

Software that is programmed into the electronic board

**GCODE**

Code that is produced using the slicing process. Describes the movements and temperature settings of your 3D printer

**After Sales Support**

☎ 1300 663 907     💻 customercare@winplus.com.au     **Model Number** HE150272     54

# Troubleshooting

**Heated Print Bed**

Surface that your 3D objects are printed onto

**Nozzle**

Opening at the bottom of the print head where filament is pushed through

**Micro SD Card**

Secure Digital memory card. This is where your GCODE files can be stored and accessed by the 3D Printer

**PLA Filament**

Commonly used plastic filament with low environmental impact

**Slicing**

Process that turns a 3D Design into a code that can be used by 3D Printers

**Spool**

Filaments of PLA and ABS are wrapped, creating a spool

**Stepper Motor**

DC motors that move in discrete steps. This allows for precise positioning and/or speed control

**.stl**

A common file format for 3D Designs/Models

**USB Cable**

This cable allows communication between the Cocoon Create 3D Printer and a computer, using the USB port of the computer

# COCOON
## create

# Appendix

## References

Some of the 3D Models used in this manual and other included materials were sourced from the below:

**Unicorn Head by kellesabelle**
Published on February 5, 2012
www.thingiverse.com/thing:17090

Creative Commons - Attribution - Share Alike

**Voronoi lamp by Markellov**
Published on December 9, 2014
www.thingiverse.com/thing:584714

Creative Commons - Attribution - Share Alike

**Twisted 6-sided Vase Basic by MaakMijnIdee**
Published on March 7, 2012
www.thingiverse.com/thing:18672

Creative Commons - Attribution - Share Alike

**General purpose hook by Erikjuh**
Published on October 20, 2015
www.thingiverse.com/thing:1081862

Creative Commons - Attribution

**Cable Holder (Cable Clip) by bardiaesm**
Published on April 6, 2013
www.thingiverse.com/thing:70549

Creative Commons - Attribution

**Twisted bracelet by Gepetto**
Published on January 25, 2015
www.thingiverse.com/thing:651677

Creative Commons - Attribution - Share Alike

**Geoflower by EMRL**
Published on August 25, 2015
www.thingiverse.com/thing:982849

Creative Commons - Attribution - Share Alike

# COCOON™
## *create*

### 3D Printer

# Repair and Refurbished Goods or Parts Notice

Unfortunately, from time to time, faulty products are manufactured which need to be returned to the Supplier for repair.

Please be aware that if your product is capable of retaining user-generated data (such as files stored on a computer hard drive, telephone numbers stored on a mobile telephone, songs stored on a portable media player, games saved on a games console or files stored on a USB memory stick) during the process of repair, some or all of your stored data may be lost. **We recommend you save this data elsewhere prior to sending the product for repair.**

You should also be aware that rather than repairing goods, we may replace them with refurbished goods of the same type or use refurbished parts in the repair process.

**Please be assured though, refurbished parts or replacements are only used where they meet ALDI's stringent quality specifications.**

If at any time you feel your repair is being handled unsatisfactorily, you may escalate your complaint. Please telephone us on 1300 663 907 or write to us at:

Winplus Australasia
PO BOX 537
Bayswater Business Centre
Bayswater, VIC 3153

Winplus Help Desk 1300 663 907
(Operating Hours: Mon-Fri 08:30AM to 6:00PM)

customercare@winplus.com.au
www.winplus.com.au

**AFTER SALES SUPPORT**

☎ (AUS) 1300 663 907          ⌨ customercare@winplus.com.au

MODEL: HE150272 PRODUCT CODE: 52914  02/2016

**1**
**YEAR WARRANTY**