

ARDUINO MATERIA 101 USER MANUAL



REVISION 02
24 NOVEMBER 2014

WELCOME TO 3D PRINTING

ARDUINO.CC/MATERIA101

© 2011-2015 Arduino LLC. All rights reserved.

The Arduino name and logo are trademarks of Arduino, registered in the US and in the rest of the world.

Other product and company names mentioned herein are trademarks of their respective companies.

Materia 101 is designed by Sharebot for Arduino.



INDEX

05	<u>Technical Support</u>	18	<u>04 – Configuring the printer</u>
		19	Loading the filament
06	<u>Technical Specifications</u>	19	Preparing the printing bed
		19	Learning to set up the plate
07	<u>Warning: Temperature</u>	19	Calibration
		20	Regulating final leg of axis Z
08	<u>01 – Preliminary Operations</u>	20	Calibrating axes X and Y
09	The anatomy of the Arduino Materia 101	20	Recovering of scraps and usage of M99 (software calibration)
09	Opening the package	20	Squaring off axes X and Y (hardware calibration)
09	The printers position	21	Putting in tension the extruder’s rubber bands
10	Assembling the spool-holder		
10	Unblocking the extruder	22	<u>05 – Our first print from a SD Card</u>
		23	SD card
11	<u>02 – The 3D printing Process</u>	23	Printing profiles
12	Model	23	Starting the print
12	Downloading from internet	23	Stopping the print
12	From a 3D model to a printing file	24	Modifying printing parameters
12	A good STL, a good slicing, a good print	24	Detaching the 3D printed object
13	Printing file		
13	Storing the files	25	<u>06 – Creating G-Code files</u>
		26	Slic3r
15	<u>03 – LCD panel</u>	26	Configuring Slic3r
16	Informative mode	26	Loading the model to print
16	The control knob	26	Basic parameters of Slic3r
16	Access to principal menu	27	Slicing
16	1° level active printer		
17	1° level printer in print	28	<u>07 – Materials and printing plate</u>
17	“Prepare” Menu	29	How ABS behaves
17	“Tune” Menu	29	How PLA behaves
		29	How other materials behave

31 **08 – Consideration about
model orientation**

- 32 Orientation
- 32 Just one object?
- 32 Support structures
- 33 Printing Details

35 **09 – Everyday maintenance**

- 36 Cleaning the printer
- 36 Printing bed
- 36 Feeding mechanism
- 36 Nozzle
- 36 Lubrification
- 36 Tightening the belts
- 36 Correct ventilation
- 37 Updating the firmware

TECHNICAL SUPPORT

If you have any problems in using our printers, the procedure to be followed is as follows:

- Check the manual and guides available on the web site;
- Contact the dealer from which you purchased the machine;
- Contact technical support by filling out the form beside remember to fill in all fields.

Our technicians will get in touch with you within 3 working days. In case your printer should re-enter in the factory to be repaired, our technical support will give as sign a file number to return and we will provide all the additional information required. Attention: the package should be the original one, penalty forfeiture of the guarantee.

Upon receiving your printer, our technicians will assess whether the problem stems from a failure, in this case the repair under warranty will be carried out for free, or by improper use of the printer or by an improper use of material not approved by Arduino: in the latter case we will provide you with a quote to request your consent to repair. For out-of-warranty printers may be required a quote for the inspection. Troubleshooting is a very important task for us and we think is necessary to share experiences, problems and solutions. Because of that please give priority to the use of the forum that will also allow others to take advantage of our and your efforts. If you are interested in customized assistance programs please contact support@arduino.cc

TECHNICAL SPECIFICATIONS

Printing technology:

Fused Filament Fabrication

Mono extruder printing volume:

140 x 100 x 100 mm +/- 5mm

Theoretical positioning resolution axes X and Y:

0,06 mm

Positioning resolution axis Z:

0,0025 mm

Extruder hole diameter:

0,35 mm

Filament to be extruded diameter:

1,75 mm

PLA best extruding temperatures:

200-230°

Tested and supported printing filament:

PLA

Experimented printing filaments:

Cristal Flex, PLA Termosense, Poliuretano Termoplastico (TPU), PLA Sand, PLA Flex, PET

External Dimensions:

310 x 330 x 350 mm w

Weight:

10 kg

Consumption:

65 watt

Electronic PCB Arduino Mega 2560 with Open Source Firmware Marlin (link to be decided).

LCD Screen of 20 x 4 with an encoder menu of navigation

Presettet printing values for PLA.

Extruder block with regulation pressure upon filament.

ATTENTION TEMPERATURE

The fusion and extrusion system of the thermoplastic filament requires a heated end up to temperatures of 250 degrees and more. The contact, even for a very limited time, with the heated part causes burns. You must avoid to put your hands or other body parts in the printing area when printing and for minutes after the printing process is over. The current temperature of the extruder is indicated on the LCD screen.

01

PRELIMINAR OPERATIONS

*Your Arduino Materia 101
is equipped with a series
of accessories that
allow you to make it
immediately operative.*

ANATOMY OF ARDUINO MATERIA 101

Your professional 3D printer Arduino Materia 101 is a tool of precision, ideated, developed and assembled in Italy by qualified staff. Before leaving the factory it was submitted to a quality control and to a quality test that guarantees its operation. It's very important, before you can use the printer, knowing all the technical terms used in this handbook and identify all the specific components that form a Arduino Materia 101.

Fig. 1:

1. Z axis carriage
2. Glass print bed
3. Clips for print bed
4. Print bed adjustment screws
5. LCD panel
6. ON/OFF switch

Fig. 2:

1. Extruder
2. Extruder cable
3. Filament inlet hole
4. Fan for cooling printed material
5. Print bed adjustment screws

OPENING THE PACKAGE

Your Arduino Materia 101 is equipped with a series of accessories that allow you to make it immediately operative; there is no software because it is open source and constantly updated. To download the latest version, please visit the Arduino website.

With care and without damaging the packaging, remove the various elements that hold the printer inside the cardboard box. Then lift the printer out of the box by grabbing it by the internal metal frame.

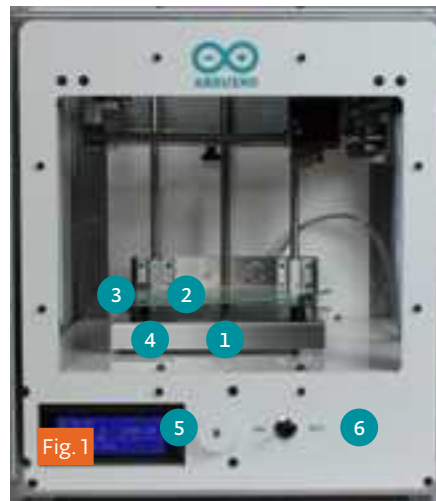


Fig. 1

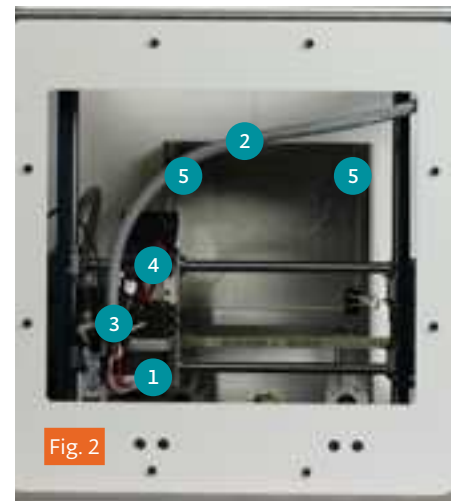


Fig. 2

IMPORTANT: Do not pull the soft black cord on the left side that goes to the extruder; it is a power cord and NOT a handle. Continue taking the rest of the parts from box and check if the parts correspond with the list below.

Warning:

Do not throw away the original package. You could use it if you have to repackage the printer or move it.

In the package you will find:

- Assembly kit for spool-holder
- Power cable
- SD Card with a few objects pre-loaded for you
- USB cable
- Spray
- Glass printing bed

WHERE TO PLACE THE PRINTER

For proper operation, the printer must work in a clean and dust-free environment with an ambient temperature between 5°C and 35°C

It is also preferable to use a location away from drafts or air conditioning.

The power switch is located on the front of the printer. On the bottom of the printer there is a air intake for the fan that cools the electronics. This must be left unobstructed.

Remember that the filament (plastic wire) is usually delivered on a spool and that this should be put on the spool holder, included in the package. The recommended position is behind the printer, on the same level as the printer, but there are alternative positions possible if there are any walls or shelves near the machine. The important thing is that the wire is easily accessible. You can even put multiple spools on the spool holder without any hindrance or prevention of the filament reaching the extruder. For this reason, the printer must be placed in a position that has at least 35 cm of space in the back or on one of the sides.

ASSEMBLING THE SPOOL - HOLDER

To assemble the spool-holder please see chapter 3, figures 9, 10 and 11 of the assembly manual. Remember that the wire must run unhindered and facing the direction of the extruder.

UNBLOCKING THE EXTRUDER

To avoid damage during transport, the extruder is secured with clamps. The clamps must be carefully cut and removed to allow the extruder to move freely. DO NOT feed the printer until you have done this step, otherwise you will damage the motors and mechanics.

Remove all the packaging inside the machine.

02

THE 3D PRINTING PROCESS

Before you start printing with your Arduino Materia 101, it is important to understand how you arrive at a finished object.

Before you start printing with your Arduino Materia 101, it is important to understand how you arrive at a finished object. This allows you not only to know what you need to do to give form and substance to your ideas, but to identify any constraints and limitations of the technology used by the printer.

In the table (Fig.1, page 17) the steps required to make successful 3D print are summarized.

MODEL

It all starts with creating or downloading a three-dimensional model of the object that you want to print. The creation is done through one of the many modeling programs available over the internet as freeware, shareware or software for a fee. There are many and each of them has different characteristics. In this respect it is worth noting that you can have different modeling requirements and this is reflected in the variety of applications. Some are more suited to mechanical design, artistic modeling, and others for architectural design.

Remember that 3D modeling is primarily used to create objects that either a machine can make or for 3D rendering of a 2D animation or picture. In the first case, the object is designed to be made in the physical world and should therefore be subject to certain limitations that are taken for granted in the "real world". For example that a cube needs 6 sides to be a cube. In the latter, the aim is to create an image or animation, nothing more: the model might then be flawless as an image, but might be physically impossible to print.

When you choose the program that you want to use for your models, you must ensure

that it is able to create a file suitable for the production of a physical object (especially 3D printing). The programs that are made for rendering and animation might not be suitable for making 3D printed objects.

DOWNLOADING FROM INTERNET

The net offers many ready-made 3D models, but there is a difference between the models that "look good" and those that "print well".

A good indicator of an object that is printable is if it is offered in the STL-format (sites that share or sell printable objects often use this format).

If there is no STL-file, it is likely that the intended use of the object is something besides 3D printing and the results can not be guaranteed.

FROM A 3D MODEL TO A PRINTING FILE

The real "secret" of three-dimensional printing is the transformation of the three-dimensional object into a sequence of two-dimensional slices that, through overlapping, reconstruct the object. Imagine a series of slices of bread: The stack rebuilds the loaf from which they were cut from. In the case of 3D-printing every slice may consist of two-dimensional shapes not connected together, because it is their addition to the layers below and above that will amount to a finished object.

Thanks to this breakdown, the objects can be as complex as desired. The printer will draw one layer after another without overlap. A layer shaped like a square or a doodle have

the same difficulty for the printer.

The postprocessing-program takes a bunch of parameters and applies them in the process of cutting the 3D model into 2D slices. This process is simply called "slicing" and programs that do this are called "slicers". To be able to do this in a proper manner the slicer needs a mesh of surfaces shaped like triangles in 3D space. The mesh has to be closed and not have any surfaces missing. The triangle faces should be correctly oriented, not confusing the "insides" with the "outsides". Sometimes the model is incorrectly made. If it is a small issue, for example a missing triangle, the program might be able to fix the issue. Otherwise, the resulting errors might prevent the correct reproduction of the object.

A GOOD STL, A GOOD SLICING, A GOOD PRINT

The STL file is the starting point for the entire procedure, so anything you do not want to print should be removed from the file (hidden objects, etc.).

The 3D drawing must be a solid, not what only appears to be a solid. Similarly, the design must be as clean and precise as possible. Articles must have an inside and an outside that are well defined so that the surfaces perfectly match each other without leaving holes. Most 3D design software gives you the possibility to carry out an analysis of borders to check for holes.

The 3D drawing must be oriented in such a way as to have a face adjacent to the XY plane of your drawing software. In other words, the design should not "float in a vacuum", otherwise the machine

will not print. If you are working on a STL downloaded from the internet or given to you by a client, there is free software for checking and correcting STL-files, such as Netfabb. The "slices" are printed in a specific format (called G-code). This format is nothing more than a set of instructions for moving the instrument (cutter, laser or extruder head) along a path (X, Y, and Z axes). Each slice contains the commands to move the extruder on the x and y-axis and extrude the molten plastic to create one layer after another and finally make a whole object. At the end of each layer, the G-code tells the machine to lower the z-axis just enough to add a new layer.

Every 3D object can be transformed with the process of slicing in a variety of G-code files, each different from the others depending on the slicing parameters being used. For example: the slices have the inside full, empty or partially empty, or the walls are made with one, two, three or more passes. Even the thickness of each layer is one of the parameters. The same object can be printed full, empty, with 100 or 200 slices and more or less robust and stiff despite all versions having a similar appearance.

PRINTING FILE

The G-code files containing instructions for moving the various mechanical parts of the printer according to the specific parameters and settings have a limited compatibility between different printers. The STL file of the template can be used by anyone with a 3D printer to create a printing G-code. The G-code file works for the printer for which it was created, but could be unusable with other printers. G-code files found on the internet might not be compatible with your

Materia 101 3d printer and might damage the printer if you try and print them.

The file also sets temperatures for the extruder and the heated printing bed, tying the file not only to the printer but also to the material being used.

As you shall see in the following pages, with the LCD Panel, it is possible to modify some parameters when printing. Arduino Materia 101 offers you the possibility to edit the G-code in order to adapt it to different materials without having to redo the process of slicing. See table on page 17.

STORING THE FILES

The process consists of the stages: modeling, creation of STL files, slicing into a G-code file and print. Save the original file of the 3D modeling program you use, so as to keep any primitives (mathematical descriptions of shapes) that compose the object. Similarly, the STL files should be stored in order to slice with different settings. Save the G-code, even in different versions for the same model, so you can repeat a print with specific features used in that particular G-code file, of which you already know the final results.

03

LCD PANEL

Arduino Materia 101 is a printer that supports both stand-alone operation and control from a computer (OS X, Windows or Linux). In both cases, the LCD Panel provides functionality for booth status and editing.

Arduino Materia 101 is a printer that supports both stand-alone operation and control from a computer (OS X, Windows or Linux). In both cases, the LCD Panel provides functionality for booth status and editing.

INFORMATION SCREEN

When the printer is printing or waiting for a job, it fills the the screen's 4 rows (composed of 20 characters) with the following information (Fig.1):

The first line contains the extruder's actual temperature followed by a zero. The temperature is in degrees Celsius.

The second line on the left indicates the current extruder-position in X-and Y-coordinates, and on the right, you see the position of the bed, or z-axis. The value is in millimeters, with two decimals to indicate hundredths of millimeters.

The third row on the left shows the Feed Rate percentage of the expected print speed within the G-code. A lower percentage will slow down the speed and higher values increase the speed. You change the speed by turning the knob on the printer when you are in the information screen.

Also on the third line, in the center, is the percentage of the file that has been read from the SD card. It starts with 0%, then goes up to 100% when the print is completed. To the right of this is the elapsed time from the start of printing, expressed in hours and minutes.

The fourth line contains messages that can be generated by the G-code file with a special command, or the firmware of the printer in specific situations.



THE CONTROL KNOB

To the right of the display is a knob which rotates with small clicks and can be pressed as a button. With this knob you can interact with Arduino Materia 101: turning it clockwise will decrease the selected value, turning it anti-clockwise will increase the selected value. During menu scrolling, rotation anti-clockwise moves the marker downwards, and clockwise moves it upwards.

When the display is on the information screen mentioned in the beginning of this chapter, the rotation of the knob changes the Feed Rate percentage, increasing or decreasing the speed of all the movements and therefore also the printing speed.

ACCESS TO THE PRINCIPAL MENU

Pressing the knob on the information screen will access the main menu. Remember that the menu displayed depends on the state of the printer. If it is waiting to run a job, it offers the menu "prepare" and while you

are printing it offers the menu "tune".

The voices and the features available in these two modes are different and it is important to know them, to avoid losing time looking for functions that are in another mode.

1st level

- Info Screen
- Prepare
- Print from SD
- Reset

1° level printer in print

- Info Screen
- Tune
- Pause Print
- Stop Print
- Reset

"PREPARE" MENU

This menu is used to set up and manage the printer for subsequent operations such as printing or shutting down.

From here you can disable the motor, set the x, y and z axis to zero, heat and cool down the extruder, manually move the different motors on the printer, and change plastic filament. In the next chapter you will use this menu to adjust the level of the printing surface.

The menu consists of:

- Disable steppers
- Auto Home
- Preheat
- Cooldown
- Move Axis
- Change Filament

“TUNE” MENU

The “Tune” menu only appears when the printer is performing a job. When you are in the information screen and press the knob, the name “Tune” will appear at the top of the principal menu.

The tune menu consists of:

- Speed
- Nozzle
- Fan speed
- Flow
- Change filament

Inside, there is an additional menu named “Speed” that allows us to change the overall printing speed. The speed is expressed as a percentage and its function is identical to the feed rate information on the information screen and can be altered in the same manner. Below “Speed”, the temperature is set for the extruded plastic, under the item “nozzle”. The value is in degrees Celsius and any change must be confirmed by pressing the menu knob.

The next item is the fan speed, adjustable from 0 to 255; with very low values the fan may not even spin. The flow value set in the G-code is expressed as 100%; increase or decrease this to adjust for unforeseen changes (such as forgetting to declare the correct filament diameter).

Concerning the changing of the filament while printing: the procedure comes from the need to replace the filament when the spool is finished, but this option can also be used to change the color of the filament.

The procedure starts with moving the extruder to the parking area, then the stepper-motor of the extruder expels the current filament-thread and the internal buzzer is activated as soon as the extruder is unloaded. When you have positioned the new wire in the extruder inlet hole and it hits the drive gear, you must press the knob to start pulling the filament and let it extrude until the old plastic is fully replaced with the new stock. When the new filament is flowing well, press the menu knob again to resume printing.

04

CONFIGURING THE PRINTER

*Your Arduino Materia
101 is a product built
with care, both in design
and in the assembly
of the machine.*

Your Arduino Materia 101 is a product built with care, both in design and in the assembly of the machine. To keep it in optimal functionality, it is necessary to compensate with some everyday adjustments. For example: due to the expansion and contraction of the motors, that become hot and cold in a fluctuating manner.

With each print, the entire machine is subject to vibrations and movements. In the long run this can loosen the four screws that hold the print bed. This leads to not having a precise uniform distance between the glass plate and the nozzle of the extruder, which then inevitably fails to apply the first layer with the accuracy and uniformity that is necessary for quality printing. Straight out of the box, the Arduino Materia 101 should be calibrated with exact factory settings, but vibrations from a long journey may have altered the factory calibration (Fig.1)

LOADING THE FILAMENT

To be able to print you must load a wire of thermoplastic into the extruder, which pushes the wire into the "hot-end" and then becomes fluid, and may be extruded. The Arduino Materia 101 has a specific menu option to load or change filament and it is located under the menu "Prepare". When chosen, it starts heating the extruder and when the right temperature is reached, you can load or unload the filament (the procedure is identical to the option accessible from the menu "Tune" described in the previous chapter).

It is prohibited by the firmware to extrude plastic when the hot-end is cold and filament should not be forced by pulling or pushing it into the extruder.



Fig. 1

To load the filament in a correct manner, cut the plastic wire end (to remove any old melted wire ends), activate the "change filament" option and make sure that the wire runs past the feeding sprocket and into the hole below (Fig.2).

PREPARING THE PRINTING BED

Remove the glass print bed from the printer by releasing the clips. Spray a thin layer of hair spray evenly over the entire surface. Now wait for the hairspray to dry completely and then place the glass on the print bed and secure it with the binder clips.

Never apply the spray inside the printer (do it in a well ventilated room, by a window, or outdoors), this might severely damage the mechanical components and void the warranty. After a series of prints or when the glass plate has bumps of dirt and residue, remove the glass and wash it in lukewarm water. Always be careful when removing the glass plate.



Fig. 2

Position the glass printing bed in the printer with the two clips, as seen in picture below. The position of the clips is not binding or final. As long as they do not invade any useful print area, you can change the position based on what you are printing. The most important thing is that the nozzle does not hit the clamps.

CALIBRATING THE BUILD PLATE

This, though initially a bit tiresome, will become a habit that you do before every print of higher quality. It will not even take you a minute when you get used to it. With experience, you will be able to take action when printing the first layer if you notice any irregularities in the plastic being laid down. In general, it is a good idea to observe the first layer of the print and take notice of the shape of the deposited material on the print surface. It should be pushed "into" the surface so that it gets a good grip and does not lose its grip later in the print.

ADJUSTING THE AXIS Z END-STOP

There is a way of adjusting the overall distance from the nozzle to the print surface. To do this, adjust the Z axis “end-stop” (a simple on-off switch). Turn the black knob in the back of the machine (see picture); clockwise to increase the distance and counterclockwise to decrease the distance between the surface and nozzle.

On your Arduino Matter 101, it is recommend that you always have at least one

inch of space on the z-axis adjustment bolt available, as seen in Fig . 3 (see section 3e Figures 7 and 8 of the assembly manual).

CALIBRATING THE X & Y AXIS

There is also, for subsequent regulations that require only a rapprochement or removal of the nozzle, the ability to adjust the Z limit, running ahead or by delaying it. To adjust the limit switch intervention Z let’s turn the black knob on bottom of machine (see picture); clockwise increase the distance plate-nozzle going to decrease the total travel of Z, while turning it counterclockwise decrease the distance plate-nozzle, lengthening the total travel of Z.

— Compensation for backlash and usage of M99 (software calibration)

If you have backlash on the Y or X axis, you will notice it by looking at any hole or circle you print. Notice the direction of the backlash, for example, if the circle appears to be flatter on the X axis (towards the front of the printer), as shown in the figure above, it means that you have backlash on the Y-axis.



The first intervention is to check the tension of the belts on the axis with the backlash, after that you can use software to correct the error. In the start of G-code of Slic3r or the slicing software being used, enter the command: M99 X0 Y0.2 Z0 E0, where “Y0.2” stands for the backlash to be compensated for on the Y-axis. “0.2” stands for 0.2 mm. Enter the value and use the Materia101_test_print.zip from our support site to see if the value inserted is the correct one to remove the backlash from the axis permanently.

— Squaring off axes X and Y (hardware calibration)

If your circles are ovals, groups of boxes are not square or if the infill of the boxes are not connecting with the outer shell, maybe you need to realign the X and Y axes.

To verify that the X and Y axes are perpendicular, you can use the file “Materia101_test_print.zip”, downloadable from our website, which contains a print of two boxes. After you are done printing the test, measure the two perpendicular sides of one of the printed boxes: if they are

still incorrect (not the same length or not perpendicular) you will have to recalibrate the X and Y axes. This procedure requires a certain dexterity and experience on the machine: If you are not sure about it, avoid it. Go to the Prepare menu, select “Auto Home”; Now unscrew the bolt that tightens the strap on the left shoulder of the machine. Then slide a single tooth of the belt over through the shoulder, forwards or backwards, to correct for un-squareness of the axis; tighten the belt and repeat the test print. See Chapter 7, image 17 of the Assembly Manual for the squaring procedure.

TIGHTENING THE EXTRUDER SPRINGS

Your Arduino Materia 101 is equipped with a spring suspended idler. This solution allows the extruder to compensate for thickness variations in the filament and give the same amount of pressure all the time.

The correct pressure from the idler allows for optimum feeding of the filament to the extruder. To check if the tension on the springs are correct, enter the “Prepare” menu and choose the option “Change filament”. Wait until the machines ejection cycle is done and insert the filament. While loading, hold the filament thread with your fingers and try and keep it from being pulled into the extruder. If there is a “slip” of the drive gear (the sprocket doing the pulling), tighten the springs until you hear a sound like a “tak tak tak”. This means that the tension is high enough.

At the same time, the tension on the springs can not be too high. This is to protect the motor when it is under stress and reduce the the risk of the motor losing steps.

05

OUR FIRST PRINT FROM A SD CARD

If you have followed the instructions in this manual, you have already set up your Arduino Materia 101.

If you have followed the instructions in this manual, you have already set up your Arduino Materia 101 by placing it on a table with the spool holder, you connected the power supply, turned on the printer, calibrated the printing bed and you have also uploaded the plot strand. Now you can proceed to your first print.

THE SD CARD

In the box you will find the plastic container with your SD memory card. On this card you can put G-code files (see Chapter 2) that you want to print. Preloaded at the factory are some sample files that are ready for printing so you can make your first print knowing that the file has been prepared with optimal parameters to get the best result.

PRINTING PROFILES

Theoretically, every G-code requires a specific material that is determined by the values set by the temperature of the nozzle, heated bed temperature and fan speeds. To print the same object in different materials, one should then create separate files.

To work around this limitation, the Arduino Materia 101 allows you to select, before printing, the outline of the material that you use. In this way, you can create a generic G-code to be printed with the chosen material simply by choosing the corresponding profile. In other words, you do not need to insert the various temperature parameters and fan speeds in the slicing program because they are set correctly when you choose the desired material profile.

STARTING THE PRINT

Press the knob again to enter the menu,

then choose "SD card menu". Choose which printing profile you want to use: choosing "use set g-code parameter" will print the object with the values set in the slicing software; choosing the PLA will print profile with standard parameters of PLA and similar (extruder temperature: 220°, 255 or full fan speed).

Once you have selected the profile, the SD card's contents are displayed like files and folders. By turning the knob, you can scroll down to the file you want. By pressing the knob you make your selection and start the print. Your Arduino Materia 101 will begin the heating procedure and positioning itself at the home position of the axes, then when it reaches the correct temperature it will start with the first layer. Check that the filament is sticking to the printing surface without being too flat or only resting like a round string on the surface. If you have successfully run the calibration G-code, you will have no problems. If you have trouble, you should stop printing and redo the calibration (Fig. 1).

STOPPING THE PRINT

While printing you can pause or abort printing. The two options are available directly from the main menu as "pause" and "stop print".

It may take a few seconds for the printer to stop after "pause" is pressed.

In the second case, the printing stops immediately and for good. The extruder then goes to the X and Y home coordinates position.

Unfortunately, the point on the object where the extruder stops can be affected by heat and might deform. A runny drop of material



from the extruder could also form where the extruder stops. So the pause function should only be used for short periods of time. Try and pause it only when the extruder is tracing the infill so that the drop or deformation is hidden inside of the object.

MODIFYING PRINTING PARAMETERS

One of the features of Arduino Materia 101 is the ability to tune a number of parameters while a print is running.

The "Tune" menu will become available instead of the "Prepare" (see Chapter 3). This allows you to adapt the selected PLA profile or the set G-code values to a different material with other behaviors and characteristics (Materials table in Chapter 7), change the temperature of the nozzle and print bed, and the fan speed.

Changing the feed rate, you can slow down or speed up the overall printing speed to handle any problem, that a certain material or model might have. It might be that

this print requires more time to cool (and should be printing at a slower rate) or the model could be achieved at a higher speed. Another important parameter is the flow rate that increases or decreases the amount of material extruded. This is to compensate for any changes in the diameter of the filament wire. The flow rate can also thicken or thin out the walls of the model.

The “Fan Speed” parameter makes it possible to adjust the airflow on the object being printed.

DETACHING THE 3D PRINTED OBJECT

When your print is finished, Arduino Materia 101 lowers the print bed and parks the extruder. Now you have easy access to the print surface to detach the finished object.

This is a delicate and inevitable operation. Because of the way the object is printed, it is necessary that the first layer adheres well and remains clinging to the plane for the duration of the print itself. If the bond was not good, mechanical stress and vibrations or even a small nudge from the extruder would detach the print and consequently ruin it.

Detaching the model thus requires a certain expertise and some tools such as a thin spatula or a box cutter (Fig. 2).

Be careful when removing the object with these tools. It is recommend to wear protective gloves to avoid cuts from the blade or the glass plate.

Try and insert the blade a few millimetres under one side of the printed piece, then with a horizontal movement, without trying to lift the piece, move the blade towards

the opposite side of the print. Usually the piece comes loose without much force and without you reaching the other side. If the model has thin leg parts touching the surface, you should detach these first without lifting them and then detach the part with the broadest base. In this way you avoid breaking the thin parts. In the case of vessels or parts with a single wide base, start on one side, trying to insert the blade deeper and deeper. Do not push the blade directly in but use an oscillating movement to advance the blade.

Avoid pulling on the printed object or using it as a lever, because it could break. If the printed object has a big base surface, you can remove the glass print plate and place it under lukewarm water to allow the hairspray to dissolve and allowing the easy removal of the object.

Remember that the extruder is warm just after the print is done. The extruder’s solid brass nozzle remains hot for several minutes at temperatures sufficient to cause a burn. Look at the temperature on the display and avoid any contact with the nozzle as long as it indicates a value higher than 50 degrees. If in doubt, do not detach the piece until the temperature has dropped to safe levels.



Fig. 2

06

CREATING G-CODE FILES

There is a wide range of programs available for modeling software and for slicing software. They come in free open source form as well as paid professional form.

There is a wide range of programs available for modeling software and for slicing software. They come in free open source form as well as paid professional form.

Slic3r is a free one and is used as a reference slicing software. Its versatility makes it useful for beginners and professionals alike.

SLIC3R

It is made by Alessandro Ranellucci and it is an open source slicing software. It can be downloaded from the download section on www.arduino.cc/Materia101.

At the moment of the writing, the suggested version is 1.1.5, which is the latest stable version. Once you have finished the download, unzip the folder and double click the icon "Slic3r". The program will run without installation. Create a shortcut on your desktop. During the launch, the program may ask you to set the data and the parameters of your printer: click "cancel" and loaded the Arduino Materia 101 profiles as shown in the next paragraph.

CONFIGURING SLIC3R

As a first step, it is recommend that you switch to expert mode. From the file menu choose "preferences" and in the drop down menu select "expert". To confirm the mode you need to close and restart the program. At this point, it is necessary to define the parameters for the printer. In order to simplify the process, these are made available on the arduino web page. They are also part of the files you already downloaded.

After that, choose "load config ..." from the File menu; in the folder "profiles" that you have already downloaded, select the file "ArduinoMateria101.ini" and, once loaded,



click on the diskette icon next to the box with the name of the profile you just loaded. You must repeat the save operation for each of the three tabs "Print Settings", "Filament settings" and "Printer settings". The profile will be available in future sessions only after saving, otherwise it will be available only for this session of "Slic3r".

If you want to create different profiles, you can modify one parameter and then save the new configuration with a new name. It is recommend not to overwrite the profile already supplied by Arduino (Fig. 1).

Note: try and keep the names of you configurations descriptive of the parameters it contains.

LOADING THE MODEL TO PRINT

In the "Plater" tab, you click on "Add" to add your 3D model in .stl format; you can also drag it into the program from an open folder. Slic3r will center the object automatically, but it will not automatically lay it down on the surface. If your .stl file has not been

created with the base of the object on the zero point of the Z axis, your print will fail.

You can lay out multiple STL's or repeat the same STL multiple times. At the beginning, it is recommend you to try one file at a time.

BASIC PARAMETERS OF SLIC3R

The Arduino Materia 101 allows you (thanks to 'Use GCODE params" and PLA profiles) to generate a G-code by setting only the geometric parameters of your interest. You can create specific G-code for individual materials. If you are only using PLA, you do not need to.

For further information, please consult the Slic3r manual (manual.slic3r.org).

— Layer height: Inside the tab, "print settings", in the menu, "layers and perimeters", you will find the item, "layer height". This is the height of a single layer and determines the printing resolution (for shapes on the z axis) and, consequently, the printing time. The preset value is 0.2 mm and is a good compromise

between speed and quality, but you can go down to 0.05 mm or up to 0.35 mm.

— Infill: In the next menu, “infill”, the parameter “fill density” expresses the percentage of fill in the printed object. You can set values from a minimum of 5% to a maximum of 100%. This parameter will affect the speed of printing. The lower the fill, the faster the printing. Obviously, with a low filling will have a brittle and light piece. For this, the fill parameter can be varied depending on the outcome you want to achieve. The entry, “fill pattern”, lets you choose the pattern that you want to fill the object with (rectilinear, honeycomb, etc.), from a drop down menu.

— Support material: In the menu, “support material”, you will be able to select the item, “generate support material”, and the program will automatically support areas with overhang (areas of the model that are suspended or sides that have an inclination of 45° or greater) with support towers. Before you select this option, you should read the details in Chapter 8 of this manual and evaluate any workarounds you might implement (changing orientation of the piece, etc.).

SLICING

Once you input the parameters of your interest, you can start the slicing process. Go back to “Plater” tab and click on “export G-code”. Then you have to select the destination and the file’s name, and the process will start. This process could take a few minutes because it depends on the parameters chosen and model’s complexity. When the slicing is over, copy the G-code file onto the SD Card: you are ready to print.



Fig. 2

07

MATERIALS AND PRINTING BED

One of the most critical elements in current 3D printers is the superficial treatment of the printing.

One of the most critical parameters of 3D printing is the surface of the printing bed. When you extrude first layer of the object, it is smeared by the brass tip onto the print surface. If the cooling plastic loses adhesion, it is very likely that the print will be damaged before the object is completed. There is no single solution for all types of filament. Because of this, we mainly focus on the two materials most commonly used; PLA and ABS, and compare these with other materials.

HOW ABS BEHAVES

It is a thermoplastic with characteristics that can be varied by changing the relationship between the three main components (acrylonitrile, Butadiene and styrene) to obtain a material with more flexibility, strength, a melting point higher or lower and even other physical parameters such as the percentage of thermal retraction/expansion. ABS is used extensively in industry with injection moulding of common everyday objects. It is also the material that the first RepRap 3D printers were using to print engineering parts and gears. It resists high temperatures before softening and losing its mechanical resistance and dimensional stability. On the downside it needs a higher printing temperature of 230 or more degrees Celsius and during cooling its size reduces noticeably, leading often to deformation of the first printed layers and the consequent detachment from the printing surface. To avoid this phenomenon, there are two countermeasures to use in combination. The first one is to have a surface covered with a material that has strong adhesion with the ABS. The second is to keep the heat of the surface (between 70 and 115 degrees) to allow the printer to build the entire piece without thermal retraction being so pronounced.

A piece that is kept in shape with these two countermeasures throughout the print will be far more dimensionally accurate with the 3d model. Often, however, it happens that there is a partial detachment on thin parts, elongated with significant coverage of the bed. In these cases, the printing is completed but the deformed parts might be arching upwards. This is called warping.

The material, which at the moment is very handy for keeping the ABS attached to the printing plate, is the hairspray. Its water-soluble components allow you to create a surface that adheres to the glass sheet and at the same time blends and binds with the ABS, keeping it firmly stuck in place. To remove the printed part, use a utility knife or spatula. If these do not "cut it" and you can not remove the piece (a testament of the good bond between the hairspray and the ABS), you can remove the glass tray from the printer and put it under water to soften the lacquer until the piece will come off with ease.

HOW PLA BEHAVES

This material is created by processing plant pulp waste rich in cellulose. It is preferred by many, over ABS, even though temperatures over 60° tend to make it soft. It prints at temperatures between 180 and 230 degrees depending on the additives that have been added to color it and modify its physical and mechanical properties. In relation to ABS, it is more environmentally friendly and emits almost no smell when printing. The surfaces of the objects printed with PLA are more reflective and transparent. Also the size fluctuation is much less noticeable with PLA, thanks in part to the lower extrusion temperature. Unfortunately, the range of temperatures in which the PLA remains elastic is wide: so if the piece in print is

not properly cooled, there is a risk that it might lose its structural integrity and appear deformed or lose all smaller details.

For optimal adhesion of PLA, spray the glass with hairspray outside of the printer and return it to the printer. For items with a base of a significant area, you should use a preheated plate at 60°. This is a good hack to add to your Materia 101. If you add a heated bed you also have to add a more powerful power supply.

HOW OTHER MATERIALS BEHAVE

The behavior of other materials can be related to PLA and ABS.

See the examples in the table on the next page.

Material	Attributable ABS	Attributable PLA	Tips
HIPS	Extrusion temperature: 235° Plate temperature 90°		Fan Off
Smart ABS	Extrusion temperature : 260° Plate temperature: 90°		Fan Off
Nylon	Extrusion Temperature 240°-265° Plate Temperature: 90°		Printable perfectly on garolite phenolics.
PET		Extrusion temperature 220°-230° Plate temperature 60°	Fan at 100%
Thermosense		Extrusion temperature 210°-220° Plate temperature 50°	Fan at 100%
Cristal Flex		Extrusion temperature 220°-240° Plate temperature 60°	Fan at 100%
PLA Flex		Extrusion temperature 230°-240° Plate temperature 90°	Fan at 100%
TPU		Extrusion temperature: 240° Not Heated Plate	Fan at 50%

Table of Materials

08

CONSIDER THE MODELS ORIENTATION

The possibilities offered by your Arduino Materia 101 are very broad, but like all tools, it has its limitations. This must be understood and various countermeasures must be implemented to deal with these.

The possibilities offered by your Arduino Materia 101 are very broad, but like all tools, it has its limitations. This must be understood and various countermeasures must be implemented to deal with these. 3D printing technology promises to create any shape but there are frequent cases where a print ends with a big disappointment.

ORIENTATION

Start with considering the orientation of the object in relation to the direction of how the printer builds it. The object is created in layers and each layer is created so that it overlaps the previous one. When a wall or a surface is tilted more than 45 degrees or some parts do not have anything beneath them in the previous layers, the first question to ask yourself concerns the possibility to rotate the model to solve the issues of tilted surfaces and free hanging details.

You might need to spin the object by 90 or 180 degrees, thus changing the portion resting on the print surface. In general, if you can find an orientation that allows a reasonable flat surface to be in contact with the print surface, you are ensuring that the print does not come loose during the print.

Take the example of a door printed in its natural position (vertical). The base would be a very small area as opposed to laying it down where the base would be much bigger and less likely to release from the print surface.

Maybe small details (such as the handle, for example) do not allow the construction of the door laying down. In this case, it is recommended that you remove the handle and print it separately. After they are both printed, attach it with a little



Fig. 1

glue in its original position. An excellent example of how the orientation can make a printable object without any problems (or adding media) can be found at www.thingiverse.com/thing:37978. It is a house-shaped lamp (Fig. 1).

JUST ONE OBJECT?

There are cases when printing the object in one piece is unavoidable but there are as many cases where you can and should cut it into several parts so that the object can print easier. An example of this that you often see on the internet is the Eiffel Tower. If you would print the whole small-scaled model of the tower in one piece, then the resulting object would be very small and have very tiny details and this might not be what you want. Besides, some of the details might get lost when printing a model in one piece. When cutting the model into two or three parts, it is possible to make a bigger print with a smaller printer and get finer details. The finished parts can easily be assembled with a drop of glue and then you have got a perfectly printed Eiffel tower.



Fig. 2

SUPPORT STRUCTURES

If plastic being suspended in mid air is unavoidable then you have to use some kind of support for this part. A bunch of tilted geese, for example, only have a very small area to start the print on. The supports are the solution that has technology developed to overcome the force of gravity and allow the creation of complex models. Another example would be suspended parts and other parts (balls of a bearing) with items that necessarily begin (moving upwards) from nothing, such as arms not placed on the hips in a figure (Fig. 2).

If the problem can only be linked to one detail, then you can create a solution in the model. Take the case of this famous model. Thanks to an enterprising user, the model was only modified with the addition of a support necessary to achieve a flawless printing. The lower part of the chin was started from an added support that is a tenth of a millimeter (Fig. 3 and 4).

In this way, a print that needs some

attention and has a critical point can still become a print with a guaranteed result.

The support is generated by the slicing software if it is enabled at the creation of the print parameters. Depending on the parameters and the shape of the model, you can create a support structure. At the end of printing, with some patience, you remove these structures and the piece remains clean and has the shape you wanted. Here is an example of an object that requires the supports accompanied by the G-code and is generated with these supports.

The supports are vertical structures under the cheekbones and on the inside of the orbits.

Keep in mind that the surfaces of contact between the support and the model will have a very low surface finish to make certain that the two areas are merged together. This will require a finishing touch to remove the backing material and create a finished look (if deemed necessary). In professional printers, supports are manufactured using special thermoplastic made by a second extruder. Later, these supports are dissolved with a special chemical or with thermal baths.

PRINTING DETAILS

When you prepare an object for printing, you define various parameters including the thickness of each layer. This value has a significant impact on the resolution of the object for all surfaces that grow upwards at an angle.

Imagine an inclined plane and mentally slice it. The size of the slices can vary a lot and if they are drawn with a fixed line width, you can open the spaces between one layer and the next. If the number of slices



Fig. 3

increase, they are thinner so the inclination is defined in a smoother manner. For this reason also, the Z-axis resolution is spoken of as the main resolution of 3D printers.

A side effect of a thinner layer height is that it takes greater amounts of time. In other words, a greater amount of layers to print equals more area to cover. On the other hand, the quality is going to be better and layers might bond better.

Each type of object has its own range of optimum thickness for printing and you will learn how this parameter interacts with the object. A tip is to choose a small but articulated shape that you can pursue with systematic experimentation. Practice with this and you will be able to see the effect of each variation. The definition on the two axes, X and Y, it is not controllable by the user. This is a product of the model's geometry, which is composed of triangles. If the amount of triangles describing a circle is low, that could show facets in place of soft surfaces and well rounded curves. More triangles bring greater detail.



Fig. 4

A larger amount of triangles means a larger job for the slicing software and if the triangles are in the tens of thousands, then they can even cause processing problems. As you can guess, above a certain level the details are no longer recreated by the printer and then they are basically useless. Or, rather, are only significant when blowing up the detail of a model. To reduce the number of triangles describing the 3D model, there are several programs and even Slic3r itself provides an option. It that allows you to define a size threshold for details to go ignored or simplifying and this will speed up the slicing of the model.

It should be also considered that the recommended minimum thickness of the walls is 1.05 mm on the vertical axes, consisting of three lines of plastic (perimeters).

09

EVERYDAY MAINTENANCE AND CLEANING THE PRINTER

*Arduino Materia 101
does not require any
special maintenance,
but like all machinery,
efficiency is at its highest
if it is kept clean.*

EVERYDAY MAINTENANCE AND CLEANING THE PRINTER

Arduino Materia 101 does not require any special maintenance, but like all machinery, efficiency is at its highest if it is kept clean.

THE PRINTING BED

The first layer of the object you print is essential to the success of the print. For this reason, the print bed must be perfectly smooth with a uniform layer of hair spray applied. After many prints, overlapping layers of hairspray together with residue from earlier printing might compromise the flatness of the glass print surface, creating irregularities. It is easy to carefully remove this with lukewarm water and applying a new surface of hairspray.

FEEDING MECHANISM

The drive gear (or sprocket) of the extruder and its cushioned pulley tend to "eat" or chew on the filament. For a perfect feed of filament, it is necessary to remove accumulated dust or other debris left on the pulley and on the drive gear.

If you want to do a thorough cleaning, unscrew the 4 lateral screws (those with springs) and sweep away residue from its bearing. After this you can reinstall everything and run the filament through the mechanism.

During the loading of the filament, try and secure the wire with your fingers; if this were to cause a "slip" of the drive gear on the filament, the springs must be tightened until you hear the motor losing a step (you should hear a sound like a "tak tak

tak"). The springs should not be too tight. Avoid putting the motor under too much strain and consequently losing steps.

THE NOZZLE

If you want to remove plastic residue on the tip of the nozzle; go to "Prepare" menu and choose "Preheat", then select "move axis" and extrude about a half of a finger length of material or until it pulls the leftover material away from the tip. Now select "cool down" and when the temperature of the nozzle is between 100-150°C, remove the dribble of plastic that hangs from the nozzle with tweezers.

LUBRIFICATION

This is required only after intensive use of the machine (or early on if it is used in a dusty environment -strongly discouraged!-). In any case, the machine itself starts to ask for oil when it starts to whine and whistle during operations

Apply a drop of sewing machine oil or petroleum jelly on the Z, X and Y guide rods. Spread the oil on the whole length of the guide rod.

TENSIONING THE BELTS

A well tuned machine should have well tensioned belts. To tighten the belts on the y-axis, adjust the screws placed on the top of the "shoulders": clockwise will increase the tension and vice versa. To tension X-axis belts, adjust the two screws fastening the y-engine on the right side of the machine: first it must be loosened in order to move the engine downwards and so increasing the belt tension. When you have the perfect tension, tighten the two screws firmly. To

tighten the belt for the x-axis the procedure is similar: loosen the stepper motor's two screws (on the right shoulder of the machine), move the engine to get the correct tension and tighten the two allen screws.

A CORRECT VENTILATION

After leaving your printer on for long hours you might notice that it is equipped with a fan to cool the electronics. It is located on the bottom of the machine, in the center of the bottom plate. You should make sure that no debris or dust obstruct this fan.

The extruder motor has a fan dedicated for its cooling. The heat might be transferred to the drive gear without this fan. This, in turn, would make the drive gear lose its grip on the filament strand.

The fan cooling the extruder motor must always be on when the machine is in motion.

UPDATING THE FIRMWARE

Arduino Materia 101 uses open firmware and receives frequent updates. Periodically check our website for updates to keep your Arduino Materia 101 up to date with bugfixes and improvements. Instructions on how to update the firmware is provided along with the firmware. If your printer is behaving strangely or does not correspond to the behaviour mentioned in the manual, contact our support service and describe the problem. It might require an update of the firmware.