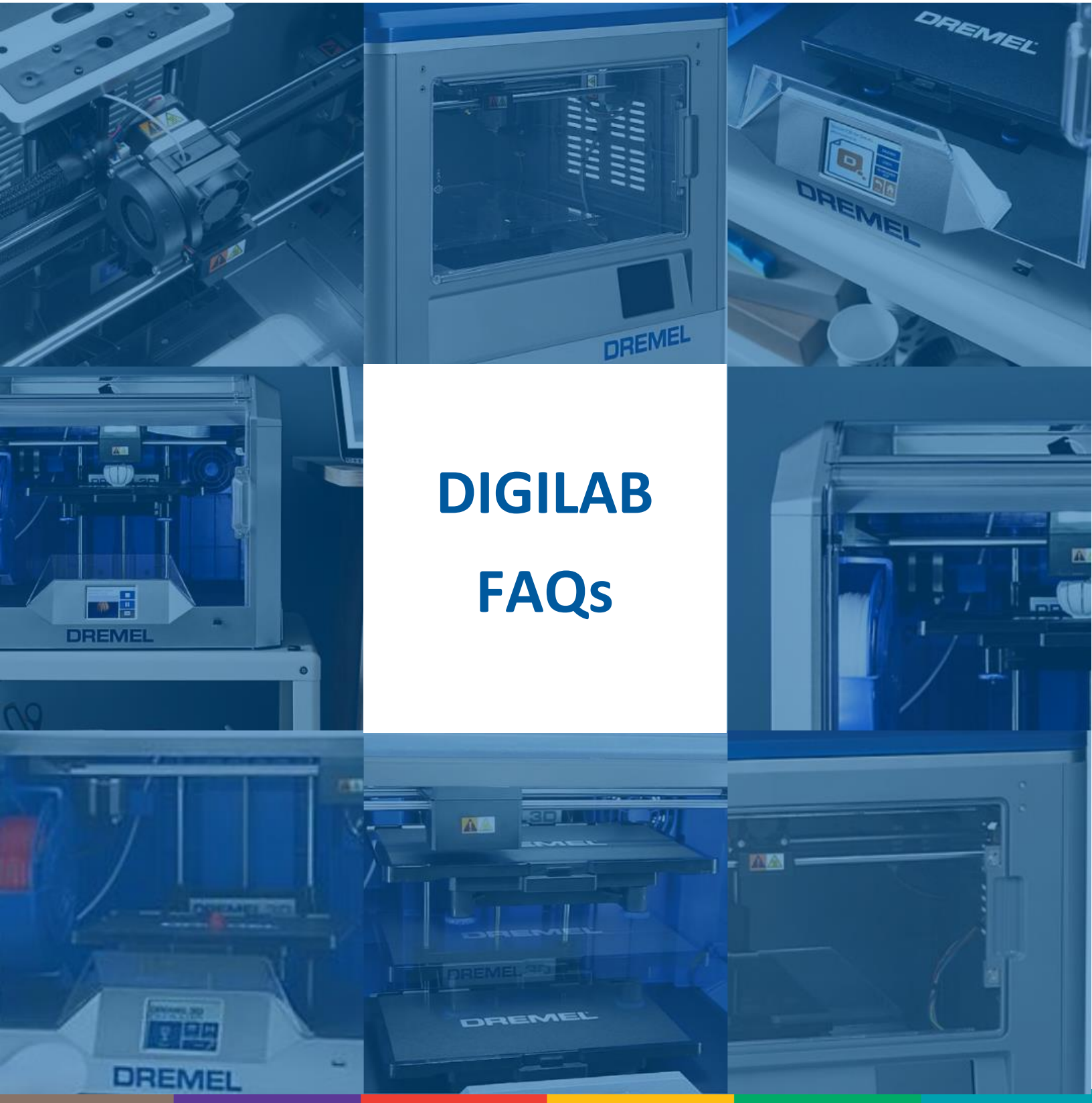
DREMEL®



DIGILAB

FAQs

INDEX

GLOSSARY of TERMS in 3D PRINTING

INTRODUCTION TO 3D PRINTING

STEPS BY STEPS

3D201

3D40-3D40FLEX

3D45

FAQs

Dremel DigiLab business

Warranty and service questions

3D Printing General Questions\Self-repair

Spare parts questions

Filaments FAQs

Files FAQs

GLOSSARY OF TERMS IN 3D PRINTING

3D printing is a method of creating 3-dimensional object from a computer aided design (CAD) file. There are different styles of 3D printing, but Dremel 3D printers use the Fused Filament Fabrication (FFF) process to extrude molten plastics layer by layer fused on top of each other, creating a 3D object.

Here is a list of commonly used terms to help familiarize you with the technology:

3D Model – A virtual 3D object typically produced in a CAD program.

3D Modeling – The act of using a computer aided design software to generate a 3D Virtual object.

3D Printer – An additive manufacturing machine that produces a 3D object by building one layer at a time.

3D Printing – The act of using an additive manufacturing machine to produce a 3D object.

Additive Manufacturing – The process of building an object one layer at a time.

Bed Temperature – Temperature of the heated build surface to aid in better adhesion of the filament.

Belt - Typically a fibre reinforced toothed gear belt, used to transfer movement from a stepper motor to a component of the printer.

Brim – A single layer printed around an object to prevent warping. Similar to the brim of a hat.

Bridge – When the printer is required to print a layer on top of a gap without supports. Just like a bridge over a stream.

Brittleness – A property of a material that will break without much deformation.

Build Plate – The surface of which the model is printed on. This can be a fixed or removable surface. Also known as the Print Bed or Print Platform.

Build Resolution – Generally refers to the height of the layer being printed - the thinner the resolution lines (or layer height) the better the resolution (visual quality).

Build Time – The time it takes a printer to complete the model.

CAD (Computer Aided Design) – A software used to generate a three-dimensional object in a virtual environment.

Chamber Temperature – The temperature at which an enclosed printer will reach during printing. Some printers monitor this to maintain a set temperature.

Chamfer – The transitional edge, typically a 45º angle, between two right-angled faces.

Clogging- Any form of disruption or complete disruption of the filament extrusion process.

GLOSSARY OF TERMS IN 3D PRINTING

Eco-ABS – A specially formulated thermal plastic derived from PLA, to mimic the properties of traditional ABS plastic.

Extruder – The components and/or assembly that performs the process of extrusion.

Extrusion – The act of pushing molten material through a die or nozzle.

Filament – A roll or coil of material to be melted during the extrusion process.

Fillet – The rounding of an interior or exterior edge of two right-angled faces.

Firmware – A software loaded onto a machine to control its functions and operations.

Elephant's Foot – Result of the first layer being squished into the bed, and/or the first layer being larger in width from settings, and/or the first layer insufficiently cooling. This causes the first layer of an object to be bigger than the following layers - resulting in the shape of an elephant's foot.

G-Code – Common name for the most widely used numerical control programming language. It is used to control automated machine tools; like CNC (computer numerically controlled) lathes, water jet tables, plasma tables, and 3D printers.

Heat Creep – Irregular heating throughout the extruder components, preventing filament from extruding and melting properly. Often causing clogs.

Hollow – An object that does not contain any infill. This will produce a fast print but may result in poor print quality and will be very low strength.

Infill – The amount of material that is used inside a model to provide strength, often referred to in percentages. 100% infill is a solid model, 0% infill is a hollow model.

Layer Adhesion – The adhesive quality between layers. Poor layer adhesion will result in separated layers. Good layer adhesion will provide a strong bond between each layer.

Layer Height – Sometimes called build resolution, it is the height of the filament for each layer. This is typically measured in millimetres or microns. (0.1mm = 100 Microns)

Leveling – The process of making the build surface (bed) as parallel with the horizontal movement of the nozzle as possible. A poorly levelled printer will give printed objects stepped sides and be dimensionally incorrect

Material Profile – Defines the settings for a specific material; such as retraction speed, retraction distance, print speed, print temperature, and bed temperature.

Melting Point - The temperature at which a solid turns into a liquid.

Micron – A measurement commonly used in 3D printing. A micron is 1000th of a millimeter; 1mm = 1000 microns, 0.05mm = 50 microns.

Nozzle – The component where the molten filament is extruded from.

Nozzle Diameter – The diameter of the material that is extruded from the nozzle. Shells and wall thickness are dependent on this diameter.

GLOSSARY OF TERMS IN 3D PRINTING

Nozzle Temp – The target temperature the nozzle requires to reach to melt the material for extrusion.

Nylon – A popular polymide material, best known for its toughness, flexibility, high impact resistance and good abrasion resistance. While being a tough material, it does have its challenges which include: prone to warping, not suitable for moist or humid environments, and requires special storage considerations. Nylon is hygroscopic and will absorb moisture; printing with nylon that has absorbed moisture will lead to many print quality issues.

OBJ file – A common standard 3D image format that uses a simple text-based structure format. The simple format may also lead to very large file sizes if the model is large or complex.

Overhangs – Overhangs occur when a new layer is partially supported by the layer below. Angled walls are considered overhangs and depending on the printer and technology, it may often require supports to print successfully. The industry standard overhang threshold is 45º.

PETG – PETG is a glycol modified Polyethylene Terephthalate (PET). The material has good surface finish, impact resistance, very minimal warping, but is prone to wear, poor bridging, and can produce thin hairs or stringiness.

PLA (Polylactic Acid) The most popular material for 3D printing due to the ease of use. It is low cost, stiff, has good strength, dimensional accuracy, and good surface finish. The downside of PLA is it has low heat resistance, it can be brittle in some instances, and it is biodegradable in sunlight.

Print Head – The assembly that moves around laying the filament down. The assembly generally includes the extruder assembly, fans, carriage, PCBA, and sensors.

Print Temperature – Also referred to as Nozzle Temperature. This is the target temperature for the nozzle to hit in order to print the selected material.

Post Processing – The act of finishing or improving the appearance of a 3D object after the printing process. This includes support removal, curing, heat-treating, sanding, tumbling, polishing, painting, gluing or fusion, etc. Nearly all 3D objects require some degree of post processing.

Print Profile – Storing user specified settings to be quickly accessed for later use. Common settings include nozzle temperature, bed temperature, print speed, layer height and supports. The biggest use case is having a profile for each kind of material.

Print Speed – The speed at which the print head moves during the printing process. This speed is typically measured in millimetres per second (mm/s). Filaments require different print speeds.

Print Volume – The largest possible dimensions of printable space. This will by printer.

GLOSSARY OF TERMS IN 3D PRINTING

Raft – A thick grid of material with a slab-like layer on top. The model is then printed on top of this slab-like layer. This aids in bed adhesion and cooling properties to help warping, but will leave a worse surface finish on the bottom of the model and requires more material to be used.

Retraction Distance – Defines the length of filament to be retracted by the extruder motor (stepper). The larger the value, the further from the nozzle the filament is retracted.

Retraction Speed – Defines the speed of the filament being pulled away from the nozzle.

Shells – Also referred to as perimeters, it is the amount of outer rings applied to a model for rigidity.

Slicer – Known as slicing software, it is the software that takes a CAD type file, and slices it into layers, applies print profiles, and saves it as a GCODE or G3DREM file for the printer to read.

Stepper Motor – A DC motor that moves in steps. The motor has multiple coils organized in groups called "phases". Energizing each phase in a specific sequence controlled by a computer, will produce rotation and very precise movements and speed control.

STL File - A common standard 3D image format that uses geometric shapes to define a 3D image. There are two formats of STL's, binary and ASCII. Binary are the most common, as they are more compact. ASCII files are easier to read and debug.

Support – Support is extra material that is printed to give a layer a surface to rest on. Supports are typically required for overhangs and bridges and can be removed during post processing.

Support Density – This value controls the amount of support used for overhangs; generally represented in percentages. A higher value will provide a better overhang or bridge but will be harder to remove during post processing.

Surface Finish –The quality or roughness of the surface of the printed part.

Travel Speed – The moving speed of the print head during non-printing, meaning no material is extruded during this movement.

Wall Thickness – The minimum wall thickness; the thinnest dimension a wall can be printed. This can vary printer by printer; a good rule of thumb is ensuring that walls are not thinner than the diameter of your nozzle diameter (0.4mm nozzle = 0.4mm thinnest wall)

Warping – Due to the high heat used in the 3D printing process and the variances in materials, materials cool down at different rates in different areas, resulting in warping. Warping can be reduced with things such as a heated bed, bed glue, build sheets, rafts, brims and selecting the correct material.

Watertightness – An industry term used to describe a complete model. Having your model surface closed/complete will be watertight.

X-Axis – The side-to-side (left and right) movement of the print head.

GLOSSARY OF TERMS IN 3D PRINTING

Y-Axis – The front and back movement of the print head.

Z-Axis – The up and down movement of the print bed.

Z-Axis Offset – Also referred to as Z-Gap, it is the working gap between the tip of your nozzle and the build surface. Since the filament needs somewhere to go when being extruded, this gap provides area for the nozzle to extrude onto the build plate. The higher the offset, the more distance between the nozzle tip and the build surface. This is adjustable and is critical for proper bed adhesion.

Z Distance – The vertical distance between two objects of whatever subject it applies to (supports and rafts).

INTRODUCTION TO 3D PRINTING

Successful 3D printing on your DREMEL 3D printer follows four steps:-

1. Designing the model you wish to print.

   Design plays a major part in the process. You can download files from the internet (resources such as www.thingiverse.com - but there are many available) or you can design the objects yourself.

   Designing objects yourself can be via entry and cost-free (at time of writing) design tools such as www.tinkercad.com. This is targeted at schools and colleges for first steps in Computer Aided Design (CAD). You can also print files designed by fully featured CAD tools used in engineering and industry.

   All you need is for your 3D designing software to be able to save files as STL or OBJ formats.

   Design tip: try to avoid areas that are 'floating' unsupported. The printer prints from the bottom up and if your design has lots of 'bridges' between items it will require more support structures to be printed. It does this automatically, but more support structures require removal but can leave marks.

2. Slicing the 3D file into a printable file.

   Load your desired STL or OBJ 3D-model into software called a "slicer" such as Ultimaker's Cura (https://ultimaker.com/software/ultimaker-cura). This software takes your model and transforms – slices – it into layers that your 3D printer can print

   Slicing tip: Always go for the standard settings if you're new to slicing as the software creates the best file automatically. You might also find that moving or rotating the object gives better results. Advanced users can explore custom settings. These give near limitless flexibility for those users with extensive experience.

3. Loading the sliced file into the printer

   Saving the sliced file into your printer via your local network, via cloud printing (requires subscription) or typically onto either an SD-card (3D20) or a USB flash drive (3D40/Flex/45). Insert these into the printer memory card slot, select the file – and print.

   File handling tip: Using the SD-card or USB flash drive is fastest as it does not rely on network connections or internet traffic. Although the DREMEL printers have USB cable sockets, these cannot currently be used to transfer data to the printer.

4. Printing the file

Before you can print the file you must initialize, or calibrate, your printer and load the filament – the reel of material on the spool - that is converted into your model by the printer. As with any 2D paper printer, you should ensure this is done on first use and then checked periodically if you notice drops in print quality.
… continued

For 3D printing you must verify the print surface is level and that the gap between the bottom of the printing nozzle and the printing surface is correctly set. Select the levelling and gap adjustment options from the user menu after you have loaded the filament.

Printing tip: The most common causes of failed prints include 'spool wrap' and 'bed adhesion'.

Spool wrap happens when the filament winds itself up on the spool which will cause the printer to 'jam' slowly until it stops. When you remove a spool or unpack a spool, always make sure the loose end of filament is held in the small holes on the rim of the spool. If you're unsure, unwind about 3 metres / 9 feet of material and rewind.

Bed adhesion problems are not uncommon in 3D printing and means that the object you're printing has not stuck to the print bed. It can simply be pushed off the print bed or wrap itself around the nozzle requiring cleaning. Always check level/nozzle gap first. Further issues can be due to design (not enough contact to base), not enough friction (not enough glue or worn print sheets) or incorrect "bed adhesion" settings in the slicer. Try switching to a "brim" in the print options and re print.

The above is by no means an extensive list of topics, but we hope this will prevent the majority of issues for those new to 3D printing.

Written by one of our internal 3D experts, at the end of this FAQ document is an article with many practical tips and observations to improve your results and how look after your machine. We suggest you read this to help shorten the time it takes to gain the experience and become a 3D expert yourself.

We thank you for your purchase and hope you enjoy using your DREMEL 3D printer.

STEPS BY STEPS

3D20

- 3D20 Cleaning Extruder Tip
- 3D20 Leveling Build Platform
- 3D20 Loading Filament
- 3D20 Nozzle Replacement
- 3D20 Unclogging

3D40-3D40FLEX

- 3D40 Connecting to WiFi
- 3D40 leveling arm replacement
- 3D40 Leveling Build Platform
- 3D40 Nozzle Gap Calibration
- 3D40 Top Cover and Run-out Sensor Replacement
- 3D40 Unclogging the extruder

3D45

- 3D45 Connecting to WIFI
- 3D45 DigiLab Slicer
- 3D45 Extruder Replacement
- 3D45 Glue Application Instructions
- 3D45 leveling arm replacement
- 3D45 Leveling Build Platform
- 3D45 Nozzle gap calibration
- 3D45 Packaging Instructions
- 3D45 Platform clip replacement
- 3D45 Print Removal
- 3D45 Replacing Runout sensor
- 3D45 Unclogging Extruder
- 3D45-3D40 leveling arm replacement
- Camera Replacement

FAQs

Dremel DigiLab business

Where can I buy a Dremel DigiLab 3D Printers, filaments and accessories?
Dremel will no longer continue selling Dremel 3D Printers or Dremel filaments. You can buy these products via *3PI.*

Why doesn't Dremel sell DigiLab products anymore?
Dremel has licenced the Dremel and DigiLab name in connection with the manufacture and sale of 3D products to 3PI Inc.

Are Dremel still producing the other tools in the range?
Dremel will continue producing all the other products in its range except for the DigiLab series of 3D products

Warranty and service questions

What is the warranty period for Dremel 3D products?
The guarantee period is 24 months and in the case of commercial or professional use, 12 months. The guarantee period begins from the time the tool is purchased from new. The starting date is the date on the original sales receipt.

How can I send my printer to be repaired/replaced under warranty?
Please visit the Official Bosch PT warranty process website:

https://www.dremel.com/gb/en/service--support/warranty

Where do I go to service my Dremel 3D Builder?
Please contact Dremel customer service department for service information:
https://DigiLab.dremel.com/support

How do I know if my printer is from 3pi or from Bosch/Dremel

BOSCH responsibility (no 3PI label)

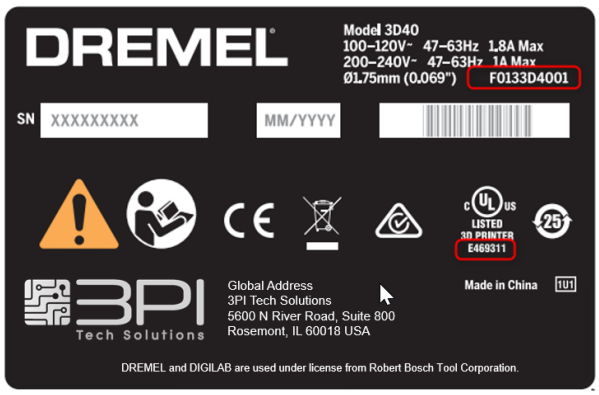
3D20

FAQs

3D40 / 3D45

3PI (responsibility of 3PI Inc)

3D40 Flex / 3D45



Are old printers out of warranty? (Chargeable repairs)

The guarantee period is 24 months or, in the case of commercial or professional use, 12 months. The guarantee period begins from the time the tool is purchased from new. The starting date is the date on the original sales receipt.

I bought my printer from Dremel, what is my warranty?

The guarantee period is 24 months or, in the case of commercial or professional use, 12 months. The guarantee period begins from the time the tool is purchased from new. The starting date is the date on the original sales receipt.

I bought my printer from 3PI, what is my warranty?

Please contact 3PI here: *https://3pitech.com/*

FAQs

3D Printing General Questions

What is 3D printing?

3D printing is a method of creating 3 dimensional objects from computer model file. Many different forms of 3D printing exist today, but Dremel's 3D Printers utilize Fused Filament Fabrication (FFF) to extrude layers of molten plastic on top of one another until an object is realized.

What is included with my Dremel 3D Printer?

3D20: Discontinued. Original included quick start guide, PLA filament (1x 0.5kg spool), USB cable, SD card loaded with model files and Dremel3D software, build tape, an object removal tool, instruction manual, unclog tool, and leveling sheet.

3D40: Discontinued. Original quick start guide, PLA filament (1x 0.5kg spool), USB cable, USB Flash drive loaded with model files and DigiLab Slicer Software, build tape (2), an object removal tool, instruction manual, unclog tool, and build sheet.

3D40 Flex: Bosch production models. Quick start guide, PLA filament (1x 0.5kg spool), USB cable, USB Flash drive loaded with model files and DigiLab Slicer Software, build tape (2), an object removal tool, instruction manual, unclog tool, and build sheet.

3D45: Bosch production models. Quick start guide, ECO ABS filament (1x 0.5kg spool), NYLON filament (1x 0.5kg spool), USB cable, USB Flash drive loaded with model files and DigiLab Slicer Software, build platform glue, an object removal tool, instruction manual, unclog tool, and build sheet.

For 3PI specifications, contact 3PI

What are the differences between 3D40 and 3D45?

The 3D45 adds several features over the 3D40:

- Support for multiple filament types. PLA, ECO-ABS, and Nylon.
- Camera for remote monitoring of builds
-Filament type and length detection
-Heated build platform to reduce model warping
-Larger touchscreen and new user interface
-Exhaust air filter, particulate and active carbon

Why have a heated build platform on the 3D45?

The heated build platform helps prevent builds from warping. This is especially true with more technical materials like Nylon and PETG.

Do Dremel's 3D Printers have dual extruders?

No, all of Dremel's 3D Printers (3D20, 3D40, 3D40 Flex, 3D45) are constructed as single extruder systems.

FAQs

How good is the build quality?

Build quality depends on a number of factors such as layer thickness and slicer settings. The Dremel printers (3D20, 3D40, 3D40 Flex and 3D45) extrude filament into layers thinner than a sheet of paper, 50 to 340 microns (0.05 - 0.34mm). The Dremel DigiLab 3D40 Flex printer has a high accuracy up to 50 *micron* resolution but you can also print very quickly with the new *Draft mode* in 340 *micron* layer height setting. The thinner the layers, the more detail can be printed but the longer it takes. Also, many slicer settings can influence the build quality, the DigiLab Slicer has settings that are optimized for Dremel's 3D Printers but experienced users can go into advanced modes and fully customize each print to suit.

How big is the build volume?

3D20: Maximum build volume is 230mm x 150mm x 140mm.

3D40 and 3D45: Maximum build volume is 254mm x 154mm x 170mm. 30% larger than the 3D20

3D40 FLEX: 254 x 152 x 170 mm build volume

How long does it take to build an object?

The time it takes to build a 3D object depends on the size and complexity of the model file and it is not an exact science. Dremel 3D model files have estimated build times for your reference.

How thick are the object layers?

The thickness of your object layers is something you can choose. The 3D40 and 3D45 Printers have a maximum resolution of 50 microns (0.05mm) for the finest or prints. The 3D20 has max resolution of 100 microns (0.1mm).

The higher the resolution, the smaller each layer and the more accurate to drawing your object will look – but the longer it will take to print.

Can I print models that will be used to store food or for medical application?

Dremel Printers are not approved for any medical application and have not been created for food/drink applications such as preparation, decoration, storage and consumption. The printers have not been tested for this purpose and is therefore not recommended. We do not recommend you to print any object that will come in either direct or indirect contact with food.

Testing policy for your DigiLab Products

All DigiLab products comply with CE Declaration and Conformity in Europe and UL Certification in North America. All products also comply with Bosch Standards.

Should I keep the packaging for my Dremel 3D Printer?

Yes. Dremel has worked hard to design your packaging so that un-boxing and boxing your Dremel 3D printer is quick and easy. The packaging will come in handy for future transportation and storage.

FAQs

How does the assisted leveling work?

The 3D40 and 3D45 Printers come with "Quick Level", which is a feature that is capable of detecting the position of the build platform with the use of smart sensor technology. It creates a reference point in the back of the build platform, then transfers that level to the front two adjustable points. It then directs the adjustment until all points are at the same level.

What is the 100% max travel & print speed for the Dremel 3D Printer?

100% print speed (60 mm/s) and 120 mm/s travel speed. You can increase both parameters but we recommend to use this speed the quality can be put at risk. The Dremel DigiLab slicer specifies the parameters and is set to produce the best results.

While my Dremel 3D Printer is in use, am I able to lock the doors?

There is no door lock or monitoring on the Dremel 3D20.

In 3D40, 3D40 Flex and 3D45: The doors are monitored, and both doors must be closed to initiate a remote print from the Dremel PrintCloud. This ensures that there are no hands inside the printer when a print is started from a user who is not near it. During any other time, including building an object, opening a door will cause a warning notice to display on the screen.

Why does my printer print an outline around the object or a line to the side of my object before it builds the model?

This is called a skirt or brim and is used to prime the nozzle to ensure smooth filament flow before beginning to print the real model. Although this can be switched off in advanced mode, it is a useful feature to see how your project is likely to go.

Will there be any marks on my print after I remove support material?

Supports are designed to be easy to remove and they should leave minimal marks on your print, however if you find some marks you can sand them with fine sanding paper. Experienced users can also adapt settings in the slicing process to further enhance the quality of prints should the standard settings not provide the desired finish.

How long does the build tape last?

Build tape is required for 3D20, 3D40 and 3D40 Flex.

3D20: The black build tape will last up to 50 prints; the black build tape can be cleaned with a mild spray cleaner, this should extend its life.

3D40: The black build tape will last up to 50 prints; the black build tape can be cleaned with a mild spray cleaner, this should extend its life. Users can also print without build tape using a 3D glue stick on the glass plate for adhesion.

3D40 FLEX: The black build tape will last up to 50 prints; the black build tape can be cleaned with a mild spray cleaner, this should extend its life.

FAQs

Is there a way to bypass the automatic setup on the 3D45?
The 3D45 chooses settings that are optimized for each type of filament. However we understand the need for adjustability. There are two levels of user control of print settings.

1. From the Main screen press *FILAMENT > VIEW/EDIT FILAMENT SETTINGS*. This will allow you to specify temperatures and speeds. These settings will be remembered for as long as that spool is loaded.

2. From the Main screen press *TOOLS > SETTINGS > ADV. MODE*. Selecting "*PRIORITIZE G-CODE SETTINGS*" will set the printer to always use the print settings included in the print file. This allows you to change parameters in your slicer and have them faithfully used by the printer.

In the build screen of my 3D40/3D45 why do some files have pictures, and others do not?

The picture information is included in the print file header and is generated by a slicer that can save files as G3DREM file types. Some slicer programs such as Cura have this as an option when the Dremel plugins have been installed. The presence of a picture will not affect the print quality in any way.

Can I let other people print to my 3D printer?
Yes, using cloud software the printer can be controlled remotely – although you must be able to ensure the build plate is clear, prepared with glue or build tape and has filament correctly installed

Can I add more than one object to the build platform?

Yes, in the step 2 "Layout" function, select "Add File" You can then add and manipulate multiple files.

What kind of filters are provided with the 3D45?
There is a particulate filter and an active carbon filter.

Do I need an extra air filtration system for my printer?
Dremel provides the ability to connect the 3D40 and 3D45 printers to ventilation systems and to fume extraction systems.

How do I use the Camera on 3D45?
The camera only operates with the online Print Cloud service. Please log into Dremel print cloud to remotely monitor your printing. You can even monitor the status of locally started prints. The print cloud is here: www.DigiLab.dremel.com/software .

Can I watch my prints while they are being made?
The DigiLab 3D45 has a built I camera. It can be viewed on the print cloud or from within the slicer.

Where is the print bed temperature measured (3D45 only)?
The print bed temperature is measured in the heating element which is in the bed floor.

FAQs

Where is the printer's filament temperature sensor located?
The temperature sensor is located in the hot end.

My new 3D40/3D45s have holes for an exhaust hose adapter. How can I attach one to my older 3D40?

The new hose adapter can be made to fit the older 3D40s with a minor modification. The hose adapter is 3D Printed.

How do I use the hose adapter on the 3D40 or 3D45?
The hose adapter is for use with a fume extraction system or for duct fan ventilation. First you will need to print the hose adapter file located on the printer. It can be printed out of any material (PLA, ECO-ABS, or Nylon). If you are using a fume extraction system, please be sure to turn it on every time you use your printer.

How do I clean my Dremel 3D Printer?
Clean your Dremel 3D printer's exterior with a lint free cloth. Clear the interior of visible debris. To avoid damaging your Dremel 3D printer do not use water, chemicals or other cleaners on the printer. Always check to ensure the extruder nozzle is clear of debris using common needle nose pliers.

If my Dremel 3D Printer is on and the power is shut off for whatever reason, will the machine automatically come back on or must it be manually "reset" for restart?

All of Dremel's 3D printers will turn back on to an unheated idle state after a power disruption. The previous print project will be cancelled automatically. If the print head is still hot, the fans will turn on to cool it down.

Is there an emergency stop button on my Dremel 3D Printer?
No, there is no emergency stop button, however, on all Dremel 3D Printers, the side power switch will immediately cut power to the printer.

FAQs

Self-repair

My printer is turned on, but it shows as offline, why?

Sometimes disruptions in the connection, like a Wi-Fi dropout, can cause the print cloud to lose sync with the printer. Try restarting the printer. Network connections differ from network to network and your network administrator may be able to assist if you are experiencing continued interruptions.

The screen on my Dremel Printer is not displaying anything? Is there anything I can do to resolve the issue?

A faulty screen means a replacement screen unit. If under warranty this can be arranged via PT ASA who will handle the proof of purchase directly with the user.

Why is my extruder clogged?

Most users will at some time experience a clogged extruder. Nozzles and hot ends are wearing parts. Typcially clogs are due to non-melting debris that gets into the extruder. That is why is it important to keep filament clean when it is open, but not being used. Clogs can also be caused by incorrect settings, using a high print temperature with a low melt-point filament, and jagged or bent ends during a filament change.

How can I unclog my extruder?

You can unclog your extruder by using the unclog tool and following the steps in the instruction manual. If that does not work, please contact customer service.

Step by step extruder repair

1. Make sure that your printer is updated to our latest firmware. Here is a link to our most recent update: https://www.dremel.com/gb/en/service--support/3d-support/firmware (different website for each country)
2. Check that the spool has not wrapped up on itself during the print. This can act as a brake and will affect the extruder from moving freely causing layer shift. We would advise you to take the spool out of the holder, unwind by a couple of meters and then rewind. Note: Always insert the loose end of filament into the holes on the rim of the reel when removing filament from the printer. This helps ensure the filament does not wrap itself on the spool.
3. Only use Dremel Filament. Opened filament of uncertain age can become brittle and causes clogs.
4. Have you conducted a z-gap calibration? This will check the distance between the nozzle and the build platform. If the nozzle is too close it can contact the object and will push to create a layer misalignment. If the gap is too large this can lead to a not well adhered first layer, leading to the object to shift. Here is our step by step guide on how to proceed with this step: https://www.dremel.com/binaries/content/assets/dremel/uk/DigiLab/how-

to-self-repair/3d45/3d45-nozzle-gap-calibration.pdf . The aim is to select a rectangle which has a neat line of filament that the first layer is adhering.

5. Are you using a Dremel DigiLab 3D45 Printer? Ensure a good layer of glue is applied to the build plate.
6. Ensure that the advanced function and gcode option Is off. You can check this via *Tools -> Settings -> Advanced mode*
7. Set the printer to auto calibrate, you will find this option via the path above.
8. Print the inbuilt Dremel frog. This will act as a quick check if the required changes has solved the issue you are experiencing or if the file you are using may be corrupt.

The extruder will not feed through any filament

Please could you follow the step-by-step guide:

https://www.dremel.com/binaries/content/assets/dremel/uk/DigiLab/how-to-self-repair/3d20/3d20-unclogging.pdf

Remove stepper motor (steps are in the clogged extruder guide) and brush away any residual filament. Take off the top of extruder and heat the assembly up this will allow for the unclog tool to reach further and to remove any filament that is very low down.

Check that the spool has not wrapped up on itself during the print. This can act as a print and will affect the extruder from moving freely. We would advise you to take the spool out of the holder, unwind by a couple of meters and then rewind. Note: Always insert the loose end of filament into the holes on the rim of the reel when removing filament from the printer. This helps ensure the filament does not wrap itself on the spool.

Why is extruder not moving to home position?

3D20: Please ensure the extruder wire tube (tube with black mesh housing) moves freely and is located away from the build platform and guide rails. If caught on the build platform, the extruder will not return to home position and a loud noise may occur. 3D40 and 3D45: Ensure that the filament is not tangled on the spool. Also check to see if the extruder is caught on the build platform or an object built on it.

Why does my printer make a grinding or clunking sound when it tries to home?

3D20: Check the wire bundle behind the extruder, it may be getting pinched between the gantry and the build platform resulting in the belt slippage.

3D40/Flex/45: A protracted noise when the extruder reaches the home position could indicate a loose and skipping belt or a wrapped spool of filament holding the extruder head against moving freely. Please contact customer service if this occurs regularly.

Why is the filament blobbing on the extruder, and how do I get it off?

This may be the result of bad filament or build failure due to poor model adhesion. If a build does not stick well to the build platform, it may be dragged around by the extruder. When

this happens the filament that is extruded sticks and forms a large blob. To prevent this, keep your build surface level and tacky (good build tape, or glue). Always check the z-gap after such an incident. To remove the blob, fully preheat the extruder, then use a pair of needle nose pliers to pull it off (be careful not to damage the brass nozzle). If there is still an issue please contact customer service.

Why does my extruder look like it is off track?
Extruder misalignment is most often caused by a tangle in the filament spool. This prevents the extruder from moving to the right. Check to make sure the spool is freely feeding filament. Misalignment can also be caused when portions of a multiple part build become loose from the build platform, these loose pieces can become wedged between the extruder and other parts of the build. If this issue happens repeatedly, please contact customer service: https://DigiLab.dremel.com/support

The layers are misaligned and are not level.
Check the spool is moving freely, increase the distance between the nozzle and build platform (Z-gap Calibration). Follow the steps in the standard step-by-step guide.

https://www.dremel.com/binaries/content/assets/dremel/uk/DigiLab/how-to-self-repair/3d45/3d45-nozzle-gap-calibration.pdf

Z-gap calibration
This will check the distance between the nozzle and the build platform. If the nozzle is too close it can contact the object and will push to create a layer misalignment. If the gap is too large this can lead to a not well adhered first layer, leading to the object to shift. Here is our step by step guide on how to proceed with this step:
https://www.dremel.com/binaries/content/assets/dremel/uk/DigiLab/how-to-self-repair/3d45/3d45-nozzle-gap-calibration.pdf

The aim is to select a rectangle which has a neat line of filament that the first layer is adhering. Once you have made adjustments, reselect z-gap calibration to ensure the best results. The rectangle with the cleanest finish should be the one in the middle of the three.

I would like to change the nozzle to a micro-swiss nozzle is there anything I have to do differently?

Replacing the nozzle with non-Dremel parts can damage the printer and in any case invalidates the warranty. Changing the nozzle only with Dremel approved parts. This is not advised unless you are an experienced user. Always check the Z gap when a nozzle is replaced.

Experienced users note when replacing the nozzle: -

1. Make sure the nozzle and extruder are as empty as possible by preheating and then clearing it out with the unblocking tool.
2. Undo the extruder cover by undoing the top bolts and one under the front edge.

3. Carefully remove the cover and disconnect the filament out detector
4. Undo the 2 stepper motor bolts to the left and right from the front. You may need to carefully move the levelling sensor to enable access to the right hand motor bolt.
5. Disconnect the motor cable but leave the motor in place as there mayu be filament trapped in this area after a blockage that prevents motor removal
6. In settings, select preheat. This will help melt filament allowing for easier removal of the stepper motor.
7. Power down the printer allowing the hot end to cool
8. Remove hot end by following instructions in this FAQ document
9. Hold the heating block in a vice and using a 9mm socket unscrew and remove the nozzle
10. Replace with a new nozzle and tighten by finger.
11. Re-install hot end with loose grub screw so it can move - you just want to be able to preheat with the hot end in place but not too tight to make adjustment difficult.
12. Preheat nozzle
13. Carefully hold heat block with adjustable wrench. Using a 9mm socket tighten only just over finger tight
14. Reset hot end fully, then tighten the holding grub screw and reassemble extruder.
15. Adjust z-gap as this will have changed following nozzle replacement
16. Verify nozzle is tight after your first print as some bedding may be necessary

https://www.dremel.com/binaries/content/assets/dremel/uk/DigiLab/how-to-self-repair/3d45/3d45-nozzle-gap-calibration.pdf

I have followed all of the recommended steps and the extruder is still blocked. What shall I do?

Extruder may need to be replaced, if this keeps occurring. You can purchase a replacement nozzle via our spare parts website:

https://www.boschtoolservice.com/gb/en/dremel/spareparts/printer-3d20-F0133D2045 (different web per country)

My print starts to warp at the sides and then the print comes off the platform and then all the plastic gets stuck around the nozzle.

Warping is caused when the plastic shrinks this causes the corners of the print to lift and detach from the build plate. This can be due to bad or no glue (3D45), worn build tape (3D20/3D40 Flex), poor quality filament, incorrect temperatures, bad nozzle gap and/or bad level. If these are all correct, it could also be a problematic STL file or a file not correctly placed on the print bed for slicing. Try printing slower, and a brim in the slicer settings (this creates more contact with the print bed) or for even more bed adhesion try a raft.

FAQs

Why is the extruder tip scraping the build platform?

3D20: The build platform has become unlevel, please re-level using the supplied leveling shim.

3D40/3D45: Accurate and fault free printing required a level bed and a correct nozzle gap to the print bed (Z-Gap). In the settings menu select nozzle gap which takes you through levelling as well as nozzle gap adjustment. Please contact customer service if you have any questions, or this does not work: https://DigiLab.dremel.com/support

The 3D45 did not detect my Dremel filament when I loaded it. What do I do?

First turn the spool approximately 1/2 turn and try the loading filament operation again. The printer will search during the loading operation. Watch the top status bar, it will display "Reading" when it is looking for the Filament. You may exit the loading operation after this reading operation completes. If it still does not detect the filament, you can select the type from the selection screen. If this problem persists or occurs on multiple spools, please contact customer service.

The build plate doesn't come off

Note: Do not attempt to remove a printed object without removing the bed first. Remove the print bed as follows:-

3D20/3D40

Carefully press the single black clip at the front of the print bed towards you to release the platform. To replace, locate the rear end of the print bed into the lugs on the printer and clip the bed into place with an audible “click” to ensure it is fitted correctly

3D40 Flex

Unclip the flexible build plate by sliding left and right the two metal clips either side of the front of the print bed. The plate is held magnetically and lifting the front edge first breaks the magnetic hold allowing you to remove the flexible plate. To replace, located the rear edge of the flexplate under the two lugs and place down on the print bed ensuring that the front edges of the build plate sit flush to allow the clips to be closed freely. If you wish to remove the whole bed completely, carefully press the single black clip at the front of the print bed towards you to release the platform. To replace, locate the rear end of the print bed into the lugs on the printer and clip the bed into place with an audible “click” to ensure it is fitted correctly

3D45

Remove the glass build plate by first pressing down the two black clips located at the either side of the front of the build platform . You should listen two clicks made by the clips. This means the clips have been released.

After the two black clips have been released, hold the platform from the two blue platform tabs located at the sides of the platform and slightly pull towards you to lift the bed out

FAQs

To replace, locate the bed under the two metal guides at the back of the printer and clip the bed down ensuring you hear two “clicks” to lock it back correctly into place.

Why are my objects getting stuck to the build platform?

Objects sticking to the build platform is a good thing as they require a firm adhesion to the build platform during the build process. This is to ensure that the object remains stable throughout building. If your prints are too difficult to remove, your nozzle may be too close to the build platform. You may need to adjust your first layer height which is available in the settings menu. Please contact customer service if you have any questions, or this does not work. See instruction manual for object removal instructions.

On my 3D45 the build is really stuck on hard, how do I get it off?

For times when the glue holds too well, we have found it best to use the provided removal tool (scraper). Work it under a corner or side and then continue to push it under the build until it releases.

Why won't my objects stick to the build platform?

There are many reasons why objects may not stick to the build platform. You should first ensure that you have clean, debris free build tape and that it is not torn or ripped. Reasons can be bad or no glue (3D45), worn build tape (3D20/3D40 Flex), poor quality filament, incorrect temperatures, bad nozzle gap and/or bad level. If these are all correct, it could also be a problematic STL file or a file not correctly placed on the print bed for slicing. Try printing slower, and a brim in the slicer settings (this creates more contact with the print bed) or for even more bed adhesion try a raft.

The print keeps shifting during the printing process, how can I avoid this from happening?

You need to ensure that enough glue is applied (If using the Dremel 3D45) to the build plate. Including a brim or skirt will also be beneficial as this is an extra layer before the actual print starts printing. Check that the filament on the spool has not become snagged – this will prevent the printer from working freely and will cause shifting prints.

Why do my objects collapse on themselves?

You may need to optimize the model design. Angles greater than 45 degrees from vertical will increasingly require external supports to prevent it from collapsing. Also, even with supports, some models may require a specific orientation to be stable while printing, or an extra layer to help the model stick. In the slicer try clicking "Generate Support" and "Build Plate Adhesion". Also verify infill percentage. If this is too low, surfaces can collapse or dip. If you are still having issues, please contact customer service for some additional ideas.

On my 3D45 when I level, it asks me to check and clean the level sensor, how do I do that?

The levelling sensor is a carefully calibrated instrument designed to measure in fine detail and is sensitive to abuse. If you observe glue residue on the level sensor, you can gently remove it with a side-to-side motion with a soft toothbrush, soft brass brush or similar. Do not use tools that are sharp and do not use force as you may damage the probe. Be careful to not touch the print nozzle behind the level sensor as it may be hot.

FAQs

Why isn't my Dremel 3D Printer starting an object?

It is possible that the model you are trying to build is corrupt, or of an unsupported type. Please turn the off then back on again and try to print one of the pre-loaded models. Also check to make sure that the firmware is up to date as there were issues with alternate file types in earlier versions.

I cannot connect my printer to the cloud

Ensure your printer is connected to the internet and that the firmware is updated. You then need to sign in and then register your printer. PATH -> Register Dremel Idea Builder -> Register Token -> Ok

https://DigiLab.dremel.com/servicedremelprintcloudbasics?action

There is no token information so I can not register my printer to the Dremel Cloud

You need your printer to be connected to your local WIFI network. Once you are connected this will generate a token which you can then use to register your printer across all softwares such as 3DprinterOS.

https://DigiLab.dremel.com/service-3d45-setup-wifi

Spare parts questions

Where can I find spare parts for my Dremel 3D Printer?

You can find our spare parts, via the spare parts website. (different web per country)
https://www.boschtoolservice.com/gb/en/dremel/spareparts/search

Firmware FAQs

How do I make sure my Firmware is the latest version?

3D20: It is not possible to update the 3D20 firmware unless you have a Windows computer running Windows 7 or Windows 2000 (KEN LARSEN to advise). If this is not possible, consider changing your slicer to Cura and download the Dremel plugin.

Connect the printer to your computer with the USB cable. Click the "Machine Information: button in the tools menu. Compare this against the latest version available for download on our website.

3D40 and 3D45: Connect the printer to the internet using the ethernet plug or Wi-Fi. The printer will automatically check every time you turn it on. There is also a "Check for Updates" button in the tools menu. If your machine is not connected to the internet, you can check the firmware version in the "About" screen in the tools menu and compare it to the version that is available for download on the website.

FAQs

How do I update my firmware if my 3D40 or 3D45 printer is not connected to the internet?

If your printer is not connected to the internet, you can download the latest version of firmware by going to https://DigiLab.dremel.com/support/3d-support

Download the version of firmware for your printer. Copy the downloaded .zip file onto an empty USB stick (with at least 70MB of storage) and unzip the contents of the firmware .zip file onto the USB stick. Insert the USB stick into the printer and restart the printer. As the printer restarts, it will check if there is a new version of firmware on the USB stick and will give you an option to install it. If the printer reaches the home screen without giving you an update message, that means your firmware is already up to date.

How can I update the firmware on my 3D40/3D45 printer?
With the 3D40/3D40 FLEX you can update the firmware via an USB stick or internet connection. It is not possible to update via the USB cable as this port is redundant.

USB Stick Instructions:

- Download and unzip the folder.

- Extract the .exe file onto your thumb drive.

- Insert the thumb drive into your printer (the printer needs to be switched off) and the .exe file will automatically begin to update the firmware on your printer.

- Once the firmware is updated this will automatically restart your machine and when the machine reboots the firmware will be updated.

https://www.dremel.com/gb/en/service--support/3d-support/firmware

Trouble updating the printer to the new firmware
A recent option is the ability to download Cura and install the Dremel plugins from the "Market Place" option within Cura. Once installed, restart Cura and select your Dremel printer

https://ultimaker.com/software/ultimaker-cura

Filaments FAQs

Which filaments can I used for the 3D20/3D40/3D45?

3D20/3D40: PLA only

3D45: PLA, ECO-ABS, PETG and Nylon

Dremel filaments
The DigiLab 3D45 printer supports Dremel PLA, Eco-ABS, PETG and Nylon. Other filament types may cause damage to the printers and void the warranty.

FAQs

- PLA is a bioplastic derived from renewable resources such as corn starch. It is plant-based and recyclable. Perfect for printing your own created 3D objects or prototypes. PLA is a thermal plastic that softens and melts when heated, building up your designs strand by strand and layer by layer. PLA has been designed to melt at a temperature compatible with Dremel 3D printers, resulting in a stable and strong object with a high-quality finish. We highly recommend using PLA to inexperienced 3D printing users.

- ECO-ABS is a modified PLA thermoplastic that provides users with durable and impact-resistant parts. Perfect for printing durable and impact-resistant 3D objects, Eco-ABS is a natural thermoplastic that softens and melts when heated, building up your designs strand by strand and layer by layer. With its RFID recognition tag, the Eco-ABS filament is designed to automatically adjust Dremel 3D printer settings by reading the filament's RFID tag, ensuring a stable, strong object with a high-quality finish.

- PETG is a thermoplastic filament that combines flexibility, strength and durability similar to nylon but is easier to print, similar to PLA. Ideal for printing flexible and heat-resistant 3D objects ensuring excellent adhesion of the layers and low risk of shrinkage and warping. PETG is a common polymer that is durable like ABS and easy to print like PLA. Its strength, flexibility and temperature resistance make it ideal for mechanical parts. PETG filament has excellent inter-layer adhesion and low risks of shrinkage and distortion, making it easy to print. The Dremel PETG filament spool is equipped with an RFID tag that allows the Dremel 3D45 to read the type of filament and adjust the 3D printer's settings accordingly.

- Nylon is a synthetic polymer that provides strong and flexible prints with great durability. Provides a very smooth surface for strong and flexible prints with high durability. We recommend that you only use Nylon if you are a professional or have extensive experience with 3D printing, as this material is more resilient but more difficult to use.

Are all of the Dremel filament types compatible with my Dremel 3D printer?
All Dremel PLA is compatible with all the Dremel 3D printers. The advanced filament types (Eco-ABS, Nylon, etc) are only compatible with the new DigiLab 3D45.

What filament material can I use?
For the Dremel 3D20 and 3D40 you should only use Dremel PLA filament. The DigiLab 3D45 supports Dremel PLA, Eco-ABS, and Nylon. Other filament types may cause damage to the printers and will void the warranty.

How long does filament last?
How long your filament lasts will depend on several factors including the print quality used, infill density, and the number of supports required. For an example, each Dremel PLA spool contains at least 175 meters of filament, and the standard Dremel Frog print requires approximately 5 meters of filament, so each spool should be able to print approximately 35 frogs.

How should I store my filament?
Storing your filament correctly extends the usefulness of the filament. Filament should always be stored in a cool, dry environment unexposed to changes in temperature or humidity.

FAQs

Always keep your filament inside the shrink-wrap plastic until you are ready to use it. Filament that is exposed to humidity or temperature extremes can become brittle or difficult to melt.

Can I get safety information for all of the filaments?
The Material Safety Data Sheets (MSDS) are available for all Dremel filaments. Please contact the service department for copies

Why does it look like there is dirt or little pellets inside the packaging of my roll of filament?

This is because the desiccant pack that comes with the filament developed a tear. This is common, and the "dirt" you see within the packaging is a form of salt that keeps the filament dry during storage. After the filament is opened, this can be safely thrown away in the garbage. Even though this contains salt, please refrain from eating it.

Is there any ABS in Dremel's Eco-ABS filament?
No, Dremel has developed a specially modified PLA blend that mimics the properties of ABS. We have avoided ABS filament due to health concerns.

Can I use more than one type/color filament on one object?
Yes. You can pause a build, change filament during the pause. The print can easily be resumed when you have completed the filament change process.

Advanced users:

1. When you have sliced the file, in preview mode locate the layer number (example XXX) at which you wish to change the filament color
2. Save the print file to your computers hard disk
3. Open the print file using a TEXT EDITOR only. Any other software will corrupt the file
4. Find the layer number in step 1 by searching for the text “LAYER: XXX” (example)
5. Insert a new line with the command “G5”
6. Save the file and copy onto a FAT32 formatted USB stick
7. Insert the file into the printers USB socket and print.
8. It will pause at the desired level for you to change filament. Note: You can only change to a filament of the same type

Why is there an RFID tag on the filament and how does it work?
The RFID Tag allows the DigiLab 3D45 printer to identify what type of filament is loaded into it. With this information, the 3D45 will set the correct temperatures and print speed to ensure your selected model prints optimally. It will even do this if the original print file was sliced for a different material. The RFID tag also allows the 3D45 to keep track of the amount of filament remaining on the spool so it can warn you if there is not enough to complete a print.

FAQs

Files FAQs

What types of files can my Dremel 3D20 use?

A recent firmware update now accepts G-CODE file extensions and this will enable the use of the Dremel DigiLab Slicer. However this is only possible for computers running older Windows systems. Newer computers should update the slicer to Cura (described elsewhere here) and download the Dremel plugin.

https://www.dremel.com/gb/en/DigiLab/software

What types of files can my Dremel 3D40/3D40 FLEX/ 3D45 use?

The Dremel 3D40/3D45 can build .g3drem, gcode and .gfiles with the use of the Dremel DigiLab slicer, Cura, Autodesk, Print Studio and Simplify 3D to create these objects .

https://www.dremel.com/gb/en/DigiLab/software

Why should I use Dremel filament?

Dremel filament is manufactured to provide the highest quality during operation and will help your Dremel 3D printer run smoothly. Also the DigiLab 3D45 utilizes specific RFID information stored on each spool to ensure the correct temperature, speed, and other parameters are used for each different filament type.

Other filaments can be used at the owner's risk as it invalidates the warranty and is only advised for experienced users.

Can I print a model that has large overhangs?

Yes, you can print such a model using supports generated by the slicing tool.n DigiLab Slicer, click the "Generate Support" button.

Is there a specific way I should position my model before printing?

The best orientation for printing usually minimizes any horizontal overhangs that would require support and maximizes the build plate contact area. These are not always the same position, and some experimentation may be required to optimize.

Can I print objects from popular CAD files (Maya, TinkerCad, Blender, Zbrush)?

Yes, you can, majority of modeling software can export files to STL or OBJ, which is the file type accepted by Print Studio, DigiLab 3D Slicer, Cura and Simplify 3D

How do I download a file from the print cloud?

Double click on the name of the file, and it will begin to download that file for you.

My model is too big to print, what should I do?

If your model is too big and resizing is not an option, you will need to break the model into smaller pieces before slicing. This can be accomplished in TinkerCad and most other 3D programs.

FAQs

Why can't I see my model file after it is loaded in the Dremel 3D software?
This could be caused by a few different reasons. A common reason is that a model file was saved outside of the build area or was scaled too small for you to see. It is recommended that you use the "Move" function to center the model and the "Scale" function to increase the model size.

Software

What software and operating system do I need to properly run a Dremel 3D Printer?

Your new Dremel printer will come with slicer software package pre-loaded on the USB drive (3D40,3D45) and the SD card (3D20). We have upgraded our slicer offerings with each printer release.

Other slicers are increasingly able to generate files for your Dremel printer. We suggest Cura as an alternative.

You can find the latest version of DigiLab 3D Slicer here: www.DigiLab.dremel.com/software

Is Dremel open source?
Dremel has utilized the knowledge of the 3D printing community by using open source elements in our printers and slicing software. All attributions are made, and the applicable code is available to the community. Please refer to you manual for instructions.

Am I limited to the Dremel Slicing Software?
Both 3D40 (latest firmware) and 3D45 will accept the following common gcode formats: .g3drem, .gcode, and .g. The officially supported slicers are: Autodesk Print Studio, DigiLab 3D Slicer, and Simplify 3D. Cura has specific plugins for the Dremel printers to ensure trouble free printing. Although code from other slicers will run, you may need to optimize the settings yourself.

Can I use the slicer CURA with my Dremel 3D printers?
Currently the latest version of CURA has a plug-in the “marketplace” tab within Cura for the Dremel range of 3D printers. Sliced files can be saved as .gcode and .g3drem for use with all Dremel printers - including the 3D20 regardless of the 3D20 firmware status. Printer profiles are also optimized for Dremel printers. When saving files as Dremels .g3drem file type, you can save an image of the print from the slicer or select a personalized one to help identify print files when in the printer display.

Can I use other 3D printer software with my Dremel 3D Printer?
Yes, if that program creates a .g3drem, .gcode, or .g file. However, the profiles included in Print Studio, DigiLab 3D Slicer, and Simplify 3D are optimized for the Dremel 3D Printers.

How do I convert 3D model files to Dremel model code?
3D models must be saved in STL or OBJ format in the program used to create these files. You can then use any of the Dremel supported slicers to convert the files into buildable models. The supported slicers are: Cura, DigiLab 3D Slicer, and Simplify 3D.

FAQs

What is Dremel DigiLab 3D Slicer?

The Dremel DigiLab 3D Slicer is Dremel's own slicing software. It is based on the open source Cura with several modifications that are specific to Dremel printers.

What is the difference between Print Studio and Dremel DigiLab 3D Slicer?
Both software programs have similar features, however DigiLab 3D Slicer is based on open source Cura whereas Print Studio is owned by Autodesk, but will not work with Windows 10 Creator or newer Window's Updates.

Can I find the Windows 10 Drivers for the Dremel 3D20, the software they were included with (Dremel3D) is no longer available on your website?

USB printing is no longer support by our software as USB support was halted at the end of 2017, therefore there are no window drivers available. Consider updating to Cura with the Dremel plugin.

How can I use my printer remotely?
You can register your 3D40 or 3D45 printers on the Dremel Print Cloud service. This allows you to upload models, manipulate, slice, start a print and monitor your print until completion. If you have a 3D45 printer, you will get a feed from the onboard camera to show its progress. The Dremel print cloud can be accessed here: www.DigiLab.dremel.com/software

How do I access online printing?
After you register your 3D Printer on Dremel3D.com, click the "Go to Print Cloud" button. You can also go directly to the printcloud here: https://printcloud.dremel.com

Once logged into the print cloud, go to the "Printers" tab, and then "Register Dremel Idea Builder 3D40/45 Printer" You will need the token/registration code from the printer. This can be found on the "About" screen in the "Tool's menu.

A new token can be requested in the "Settings" menu. For 3D45, you can also scan the QR code from the "Settings" -> "Token" menu.

How many printers can I manage on the print cloud?
There is no limit to the number of printers you can manage through your Print Cloud account.

Why does my model appear red on the Dremel 3D software build screen?
A model will be red if it is too big for the build area, floating above the build platform, or part of the model is positioned outside of the build area. Use the "Scale" and "Move" functions in the Dremel 3D software to adjust the model so that it is within the build area.

My model does not have a "closed surface", what should I do?
Dremel 3D slicer does not have the ability to repair models.

FAQs

Glue FAQs

Why do I have to use Glue on my 3D45?
The supplied glue is the best, safest, material we could find that would work with all filaments supported by the 3D45. It provides a strong adhesion that helps prevent builds from failing.

Note 3D40: You can also use glue on the glass bed of the 3D40 without build tape

How often do I need to clean the glue off of the build plate?
The glue on the build plate does not need to be cleaned every build. You can scrape rough edges off after a build, and applying more glue. This can work well once you build up a few layers. You would need to wash it if there are deep gouges in the glue layer, or it has built up unevenly and make the build platform unlevel.

How do I clear glue residue from my models after I remove them from the build platform?

You can rinse it in water or wipe it with a damp cloth.

How do I clean a 3D45/3D40 glass plate build platform with dried glue on it?
Remove it from the printer and rinse it under warm water. The glue will dissolve. After the build platform is clean, allow it to dry completely before reinstalling it into the printer. Do not use any detergents or cleaning fluids as these may prevent models sticking properly on the glass plate.

PRACTICAL TIPS FOR BETTER RESULTS

Practical tips from our experts to help new users produce better results and to look after your machine

This article is to not discourage you but help understand all the aspects of 3D printing that will apply to everyday users.

3D Models

Having a good understanding that digital design is very different from a 3D printed physical object may alleviate many pain points of 3D printing, especially to the beginners. In CAD style software, no outside elements interact with the model such as gravity, temperature, humidity, etc. While anything can be designed, it may not be easy to 3D print or provide you a good result. An example could be, if you designed a 5cm tall bush for a dollhouse project and you designed every branch and leaf in detail. Once you print it, it may end up being a ball of filament.

Below, we will cover many topics about things to consider when designing a model. All of this may even apply to models found on the internet. Gaining this knowledge will take away many frustrations and put you in a better position to get your project complete.

Overhangs and Bridging

In FFF style printing, material cannot print in thin air. Every layer must be laid on top of something, whether that be the previous layer or supports. Every 3D printer handles overhangs differently. The standard threshold is 45º. While some printers can print overhangs up to 70º, it is recommended to find a printable overhand test file to know how your printer performs. Limiting the amount of overhangs on your model will result in better surface finish. Using supports on overhangs typically causes roughness on the surface. Finding that balance of when to use supports or not, will depend on your printer and use knowledge. In the pictures below, you will see surface A will need supports and surface B may or may not need them due to the angle.

Bridging is when material is laid across two vertical points. Bridging will generally produce sagging filament unless supports are applied to hold the bridged section. Different model printers and slicing software can handle varying lengths of bridging. Running a printable bridging test will give you a better idea of the distance your printer can handle and show you the outcome as well.

PRACTICAL TIPS FOR BETTER RESULTS

Chamfer and Fillets

Chamfers and fillets are corner finishes to eliminate the sharp angle. Chamfers provide a flat transition, typically at a 45º angle and may be referred to as a bevel in other industries. Fillets provide a rounded transition of an edge.

Wall Thickness

Knowing that there are limitations to the minimum wall thickness on a model will lessen the head scratching moments. The easiest way to remember wall thickness is to never go thinner than diameter of your nozzle tip, this will keep you in the safe zone of thin walls. Having a wall thickness that is too thin will give you mixed results depending on the slicing software used. While some software allow you adjust the behavior of the software when it comes to thin walls, most software do not. Generally, if a slicing software runs into too thin of a wall, it will simply ignore and remove it. If you were to design a box with a wall thickness of .6mm, but one wall had a .3mm wall thickness, the software may erase the .3mm wall. Leaving you with an open box. This also applies to thin details in a model, not just walls.

Warping

Warping occurs to materials after it is melted, then cooled. Once a material is melted and laid onto a build surface it starts to cool down. During the cooling process, the material tends to shrink and loses contact with the build surface. Thus, resulting in the warp. There are many ways to combat warping, but we will focus on design and slicer settings in this section. The best way to combat warping is to use a material that is not prone to warping. PLA warps far less than Nylon would.

Think about the model you are building. Can you incorporate radius edges to eliminate the flat edge of the model? Having a curved edge is a good practice; this will reduce the likelihood of warping. Consider a different orientation of the model while printing as well. There are other things like rafts or brims that can aid in reduce the tendency to warp. Having a well-calibrated machine is a starting point for most users, but the real starting point is the model itself.

Level of Detail

Designing a highly detailed model is an achievement on its own, but the harsh reality is that it may be too much. Having very small details can get lost during printing. When designing a model, you need to know the limitations of your printer. Layer resolution, nozzle size, print speed, slicing software used and quality of the printer are the starting points to finding out those limitations.

PRACTICAL TIPS FOR BETTER RESULTS

Holes, Corners, and Elephant's Foot

Holes will nearly always result in being slightly undersized than the planned dimensions. This is due shrinking material upon cooling and how the layers are put down. Vertical holes (up and down) will be undersized due to the compressive nature of layers in FFF style printing. Each layer is essentially pressed into the previous one for layer adhesion. Horizontal holes will sag at the top of the hole and even will with supports in some instances.

Corners and edges of printed model will always have a radius edge to some degree. This is typically equal to the radius of the nozzle tip diameter. No model will be perfectly square. Think of using a piping bag to put a message on a cake. No matter how exact you move that piping bag in a 90º angle, the icing will not be an exact 90º angle.

Elephant's foot occurs when the first layer is squished making it wider than the following layers. This happens in varying degrees on all FFF style printers. To avoid the impact of this would be to radius or chamfer the bottom of the model, increasing the Z-Gap of your printer, adjusting your first layer thickness, or cutting/sanding off during post processing.

When to Split your Model

In some cases, it makes more sense to split your model in half, rather than having too many supports. Or, if your model too large for the print volume of your printer. You could simply cut the model in half and print two pieces and glue/melt them together. Consider printing a sphere, it would take a lot of supports to hold the sphere in place and would cause a lot of rough surface finish. If you were to cut that sphere in half, you could print both sides flat side down and utilize no supports.

Layer Direction

FFF printed parts tend to be weaker in one direction due to layer orientation. The breaking usually happens in the direction of the layer. Knowing your design purpose and use should dictate the orientation you will print. Think of trying to print a model of a pencil. While it can be printed in either direction, laying the pencil down will produce a stronger part.

Understanding What Material to Use

Choosing the right material for your application can be overwhelming at times, due to the amount of different materials out there. Material type can also dictate the level of detail you can use. Filament materials print drastically different from one another. Let us say you are looking to print a gear for a project, while the ease of printing and great surface quality is appealing in PLA, it would wear out very fast and/or break from being a more brittle material. Nylon would be a better choice in this application. Nylon provides some flexibility, impact resistance, and self-lubricating properties. However, nylon is much trickier to print with than PLA. On the other side, if you wanted to print a figurine of a dinosaur for your desk at work,

nylon would not be the best choice due to it being trickier to work with. PLA would shine in this application as it is the easiest to print with and provides a quality surface finish. Since the figurine would just be sitting on a desk, we do not need to use a strong material.

File Resolution

In some CAD programs you can alter and/or select different file resolutions for the particular file format you are exporting in. The most standard format for 3D printing is the STL format. We will skip over the differences between a binary STL and ASCII STL, as the differences do not apply to this topic so much. STL files consist of many triangles orientated to make a 3D object. The resolution of your file will determine how many triangle it uses. For instance, low resolution files will generate bigger triangles resulting in visual triangles on the finished model. This will reduce the file size dramatically. Higher resolution files will use smaller triangles that will blend more smoothly, resulting in a smoother surface of the model. The file size of the model will become much larger as due to the complexity of all the triangles. Some slicing software may not be able to efficiently generate very high resolution files. Most common slicing software will be able to handle what the everyday user will throw at it though.

Watertightness

When designing a model, you have to consider its watertightness. Having a model with an open surface and/or internal faces, even if undetectable to the eye, will most likely prompt the slicing software to warn you that your model is not watertight - although not all softwares will give warning. In some cases, the model will not slice properly and cause a bad print file.

Slicers

Not all slicing software are created equal. Each are created differently to have separate features and unique qualities that alter the look of the print and the user experience. Slicers are typically free or paid for, while some offer free basic features and then offer premium services or upgrades. The free software is usually an open source software. Meaning, it was constructed using computer code that is available to the public for use or alteration. Paid for software can have aspects of open source code, but most develop their own and provide a premium experience or result.

While some slicers share similar aspects, the typical part that differs is the slicing engine. This is the workhorse of the software. This is what converts your CAD file into a machine (your printer) readable file. Some are more proficient than others, but they achieve the same thing of providing a buildable file.

PRACTICAL TIPS FOR BETTER RESULTS

Slicer Settings

When it comes to slicer settings, no settings are perfect for every model. This holds true on any slicer software. Having optimal settings will depend on the model, material and experience. Many believe that the printer is what determines all your quality, but that is simply not true. Having slicer settings tuned for your printer is what provides you the best quality. While mostly all 3D printer manufacturers provide pre-made printer profiles for a specific unit that does not mean those settings work for every model. You may also find that the keyword in 3D printing is balance. Finding what works for you is a learned process. Below we will go over the most common settings that are changed for each print. If you need more expert tuning help, please call us at NUMBER HERE. Additionally, our Digilab slicer provides helpful tips and explanations if you hover your mouse over a slicing setting.

Quality or Layer Height

This can also be referred to as print resolution. This is the height (thickness) of each layer. The thicker the layer, the lower the resolution. Higher resolutions provide a thinner layer. So, why not just print in high resolution every time? Honestly, higher resolution takes longer, a lot longer. If you were to print a model at 0.3mm layer height (resolution) and it took an hour to print, your 0.1mm layer height will take nearly 3 times longer. Making your one-hour print turn to three hours. The printer is laying three 0.1mm layers versus one 0.3mm layer, which is why it takes so much longer. Additionally, when printing at a higher resolution the print speed is also reduced. Thus, resulting in the print taking even longer. Finding the balance that works for you is recommended. Lower resolutions will provide more visible layers and rougher surface finish, while higher resolutions will reduce the visible layers and provide a smoother surface finish. The most common layer height is 0.2mm because it is right in the middle of the spectrum.

Infill or Fill Density

Infill is the amount of material used inside your model to provide structural support, providing strength to your model. Density is referred to in percentages. The higher the percentage, the more material is used to provide strength and support for the outer walls (shells). This will also add print time. The goal is to use as less material as possible when 3D printing to reduce cost of ownership. So finding the right percentage of infill for the model you are building is recommended. 10-30% is the most common infill density for something that sits on a shelf or desk for display. 30-50% for workable low stress applications and 50-100% for workable high stress applications and/or structural components.

Supports

Using supports are critical for providing a surface for an overhang, or stability to a model with low build surface contact. Knowing when and how to use them, will be elemental in model printing success. Supports vary from slicer to slicer, in style and function. While all slicers generally provide support functions, some do a better job at surface finish and removal than others. Many slicers also provide different types of supports altogether. This provides more options depending on the model.

PRACTICAL TIPS FOR BETTER RESULTS

There are two important settings to understand when it comes to supports. Density and Z Distance. Density is the amount of support material needed for the model and desired surface finish. Higher density amounts will typically yield a better surface finish, but be more difficult to remove and require more material. Using a smaller density will generally be easier to remove, but provide a rougher surface finish.

Z distance is the distance between the top of the support and the bottom of the model. The more distance, the easier it is to remove. However, more distance can result in rougher surface finish and/or failed part. So a balance is needed for desired results.

Shells or Perimeters

Shells or perimeters are one of the critical elements to give you a strong model. Typically, the more shells you apply, the more rigid the model. Most slicing software amount this to counts. One count is equal to one shell or perimeter. Again, the amount of shells used will affect time to build and material cost. The more shells used, the more material and time to build.

Print Speed

The speed at which the extruder lays down filament. Speed is referred in millimeters per second (mm/s). Optimal settings will depend on the model being printed, print resolution, type of filament, and manufacturer of filament. While the manufacturer of the printer generally sets the print speed in the software, it may require tweaking if you use another brand of filament. Having the wrong print speed and nozzle temp could drastically reduce print quality. Typically, the slower the speed the better the print quality, but there is a point of diminishing returns. Slowing the speed down will also result in longer print times.

Bed Adhesion

This can be the most frustrating part to 3D printing for most users. While many factors come into play for bed adhesion like Z-gap, bed leveling, build surface material, bed temp, chamber temp, and the material itself. For this portion, we will focus on how the slicer can help with bed adhesion.

There are two common types of bed adhesion helpers in the software, rafts and brims. A raft essentially builds a grid with a flat top on it. Similar to a log raft. This provides many functions, the first being forgiveness when it comes to not having a perfectly flat or leveled surface. It also provides added cooling properties. This also acts as a sacrificial layer for warping problems, by providing a larger surface area on the build surface and letting the raft curl on the corners, not the model. The downside to using a raft is the bottom layer of the model will have a rougher surface finish along will using a lot more material. Some of the surface finish can be reduced by adjusting the Z distance as well.

Brims are a lot simpler but effective. A brim just adds more surface area contact to the build surface. It does this by making only the bottom layer larger (wider). Think of a flat brimmed top hat. Your model is just the centerpiece of the hat (a cylinder); by adding the brim, you are creating more contact area to prevent the cylinder portion from lifting. Brims are adjusted by counts or distance. One count equates to one nozzle width of a brim, or by distance in

millimeters from the outer edge of the model. You can also set how many layers high you want your brim to be. Brims also use significantly less filament than rafts but more than skirts.

You may also see skirts. These do not directly interact with the model, but rather prime the nozzle to get filament flowing before the model starts to print. That way when the model starts, you are providing all the amount of filament possible for the first layer of the model. Since rafts and brims start before the build of the model, they provide the same function as well.

Bed Temperature

Along with having bed adhesion helpers in the software, the bed temperature provides a good solution for adhesion. Many materials require a heated bed and/or recommend a heated bed. This is to keep the material from cooling at different rates for the first few layers. Allowing the model to build completely before cooling the build surface gives the model rigidity to cope with the cooling shrinkage. Additionally, if you are using a glue of some sort on the build surface, the heat allows it to remain tacky to hold the model down. If your printer has a heated bed, these settings can be adjusted in the slicing software. Be aware, there are working ranges for most materials. If your bed temp is too hot, it can deform your model. If the bed temp is too low, it has no effect at all.

Nozzle Temperature

Most manufacturers of filament give a range of nozzle temperatures at which to print, due to the chemical makeup of their filament. The reason for the range is that some printers do better at different temperatures due to their extruder design. Using third party filaments, you may see one manufacturer list their PLA range 180º-200ºC, while the others are closer to 220º-240ºC. Those are drastic differences when it comes to 3D printing in PLA. Therefore, understanding what temperature your filament needs to be at is critical to producing a quality model and reducing the risk of a clogged extruder.

Filament

As you may have gathered already, filament choice is another factor to consider. In this section, we will not cover every filament type and their properties, but we will recap portions of what has been said already in this article.

No two filaments are the same, even if they are the same material type. Manufacturing processes and formulations will change from each manufacturer. Some better than others and some cheaper than others. Either way, you should consider treating each roll of filament as its own.

Knowing what filament to use can be overwhelming on the surface, but relatively easy to narrow things down. There are many comprehensive guides of types of filament on the internet, but I will use four types as examples; PLA, Eco-ABS, PETG, and Nylon.

PLA is the most common type of filament for 3D printing due to its ease of use, low prices and environmentally friendly. It is made from cornstarch, which makes it biodegradable. While PLA is easy to use, it has its drawbacks. Being biodegradable in sunlight, relatively

brittle, and wears down quickly as a workable part. Most people use PLA for showpiece types like models, figurines, visual aids to a design, cosplay props, etc.

Eco-ABS is not ABS, but rather PLA that is specially formulated to mimic the properties of ABS. It shares all the properties with the added benefit of more flexibility and less prone to brittleness. This is a good substitute if PLA just doesn't quite fit the bill. If you are looking for a little bit strong PLA, this may be the choice for you.

PETG generally falls in the middle of most materials. It is made up of PET resin. The "G" stands for glycol which is an additive to make it less brittle and more pliable for printing. It combines the strength of Eco-ABS, some flexibility of nylon with the ease of printability. Due to its UV properties, it is a good choice for in sun applications. Many people use this filament for functional parts or protective covers.

Nylon tends to fall in the category of an engineering type material due its properties. It tends to be very strong, durable, flexible, and has a low friction coefficient. The downside for this filament is that it can be difficult to print. The biggest factor is warping and stringiness on high travel movement models. Given that this material is more expensive than the others are, this should not be your first choice of material for non-functioning parts. Many people use this material for protective components and working parts, like gears, levers, and covers.

Post Processing

One of the many untold truths about 3D printing is post processing. After all that design time and print time, there is more work to do. Although, the bulk of this guide is to help minimizing the learning curve and post processing time. While many objects do not require post processing, many do. Let us cover the most common ones.

Support removal is the most common form of post processing, as many models require supports. Some consider removing supports to be an art form, but it is not. With proper settings in your slicer and/or the right slicer, removal can be quite easy. The most used tools for removing supports is an assortment of small pliers and a modeling knife. With these tools, you should be able to conquer any support material thrown at you.

Sanding is another way of post processing. This can take all the small imperfections and smooth them out create a near perfect model. But, sanding may produce more work as well. Since you are sanding plastic, some plastics tend to turn milky or white. If that is not the result you need, you may have to resort to painting as well.

Painting a model takes some trial and error. Some plastics will absorb the paint, while others will special paints or primers. People also use fillers to close gaps or blend imperfections. Some fillers may require certain paints as well. There are many resources online to help determine what is needed in your scenario.

Gluing parts together may seem easy, but it can be a little complicated. While some materials just require a chemical and a little material mixed together to create a slurry paste, others may use epoxy or super glue. The type of super glue matters, since chemical make-up of the

glues vary and you are using on plastics that have their own chemical properties. The 3D printing industry has provided glue solutions for specific plastics as well.

Welding is another common practice of post processing. There a two versions of welding that get used the most and that is chemical and pen welding. Chemical welding is using chemicals to essentially melt the two connecting pieces together. This is similar to using glues that create this effect. Pen welding is unique. It requires a 3D printing pen. Instead of using the 3D pen to as it is intended (creating 3D figures), you are using it as a welder by melting like material to bond the two pieces together. This has been proven a very strong option.

Printer Maintenance

General Maintenance

Maintaining your printer is vital role in the overall success of your 3D endeavor. While the word maintenance seems like a lot of work, it really is not. The best scenario is creating a routine maintenance schedule similarly as you would your car. Since everyone uses their printer at different rates, we cannot say every week or every month do this task. Most printers count usage hours, which will aid in your scheduling calendar. Environment also plays a huge role in your schedule. Office environments may require less frequent maintenance than a wood shop in a classroom. So it is highly recommended to pay attention to the printer and adjust your schedule as necessary.

Level your printer. It is a good practice to level your printer before each build. Whether you perform the assisted leveling or the automatic leveling, it should be done before each build. This will always ensure the best possible scenario for your build and your printer. Failure to not having your bed leveled will lead to failed prints and possibly damaged components.

The Z-Axis offset (Z-gap or Z-offset) tops that list. This should be checked regularly. It is simple to check, just watch the first layer of your build every so often. This will tell you if it is still within an acceptable limit. The ideal setting will provide a nice even bead of filament on the build surface. It will look similar to a flat shoelace. If the Z-axis offset is too tight it will cause premature maintenance in the form of needing a new nozzle or requiring you to clean the gear that feeds the filament regularly. This will look like dots or splotches on the build surface. Similar to someone using a hot glue gun, squeezing, and releasing the trigger constantly. Worst-case scenario is filament not coming out at all, which you will typically hear a knocking noise. Having it too loose will cause builds to stick to the build surface, provide a stringy bottom layer, or cause buildup of melted filament on your nozzle that can lead to extensive damage if not monitored. The bead of filament will look like an oval shoelace or just a string on the build surface.

Clogging is inevitable in a 3D printer, but a routine maintenance schedule and proper use will nearly eliminate clogging. Clogging can happen from several sources; bad filament, printer not leveled, Z-axis offset not calibrated, buildup of filament on extruder gear, worn out nozzle, and improper slicer settings. The biggest culprit is typically a worn out nozzle. PLA

can typically wear out a nozzle in about 800-1200 hours of use, depending on the color and/or manufacturer of the filament. So it will need to be replaced at some point.

Cleaning your printer will go a long way to the longevity of your printer. Dust, hair, filament, and dirty filters can cause problems. It is recommended to regularly check your unit for any of this on working components. Simple dust in the air can accumulate on the guide rails and gunk up inside the rail bearings, causing bearing failure. Simply cleaning off anything you see on the rails or stepper lead screw will help tremendously.

Oiling and greasing should also be part of your maintenance schedule. While the need to oil and grease is not necessarily done visually, it can be done by feel. Running your finger across the guide rails, you should feel slickness to it. If you rub a dry finger on a rail and it provides resistance to your finger, it is time to grease or oil. We recommend sewing machine oil on the rails and white lithium grease for the Z stepper screw. Oil should be applied a drop at a time. Put a drop of oil on the rail and work it in. The rail should not show signs of liquid (the oil) on it. A little goes a long way. The grease on the other hand will show on the lead screw. Place a dab on the rail and work the surface of the lead screw. It will not take much, but the idea is to make it slippery.

The build surface also requires a degree of maintenance, whether you use glue or a build sheet. For glue applications, it is recommended to clean the build surface after every print by running under hot water and wiping with a towel and then applying a new layer of glue. This can be stretched out to every 5-10 builds with experienced users. If you are using a build sheet, it is imperative to keep your fingerprints off of the build sheet. The oils from your skin will rest on top of the surface drastically reducing the sheets functionality. In the event that fingerprints or other debris make it onto the surface, you correct it with a few tasks. Start off by cleaning with an isopropyl alcohol and allowing drying/evaporation time. Then perform a large surface area print so the filament can remove the rest from the nooks and crannies of the sheet. It may take a few builds to get the holding properties back for the sheet.

Final Thoughts

With all this learned knowledge, you will be a 3D printer guru in no time.

3D printing has a tremendous learning curve for most users, but we designed our units to take away the biggest pain points. We also wrote this article to provide you knowledge that cannot be addressed by a physical product.

Nothing will take the place of experience in 3D printing but knowing all this information provides a solid foundation to build on with experience.

Enjoy your DREMEL 3D printer as much as we enjoyed developing it.