

3D PRINTING GUIDE



3D PRINTING GUIDE

in english & deutsch

Antonia Gwendolyn Dönitz
1st part of master thesis
textile & material design
kunsthochschule berlin weißensee

mentoring
Prof. Dipl.-Ing. Christiane Sauer
Prof. Dr. Jörg Petruschat
Dr. Lorenzo Guiducci

external mentoring
Prof. Dr. Christina Völlmecke

involved institutions:
MoA Design Studio
SVFS Group TU Berlin

Berlin, Germany
summer 2024

**Hiermit versichere ich, dass ich die vorliegende Arbeit selbständig verfasst
und keine anderen als die angegebenen Quellen und Hilfsmittel verwendet habe.**

**I certify that I have written this thesis independently and that I have not used
any sources or aids other than those sources and aids other than those indicated.**

Antonia Gwendolyn Dönitz

Textil- und Materialdesign

Matrikelnr.: 7215

CONTENT

004 INTRODUCTION

006 ADDITIVE MANUFACTURING

008 3D PRINTER MANUFACTURER

014 STRUCTURE 3D PRINTER

018 STRUCTURE PRINthead

022 STRUCTURE PRINTED OBJECT

026 MATERIAL FILAMENT

030 CAD MODELING SOFTWARE

032 SLICING APPLICATION

034 PROGRAM OVERVIEW

036 3D PRINTING GUIDE

038 1. CAD TO STL

044 2. STL TO G-CODE

052 3. SETUP PRINTER

054 4. BED LEVELING

056 5. ADHEASION

060 6. PREPARE PRINTBED

062 7. START PRINT

064 8. POST PROCESSING

066 SLICER PARAMETERS

086 TABLE OF FIGURES

092 LIST OF REFERENCES

INHALT

005 EINFÜHRUNG

007 ADDITIVE FERTIGUNGSVERFAHREN

009 3D DRUCKER HERSTELLER

015 AUFBAU 3D DRUCKER

019 AUFBAU DRUCKKOPF

023 AUFBAU DRUCKOBJEKT

027 DAS MATERIAL FILAMENT

031 CAD MODELLIERUNGS SOFTWARE

033 SLICING-SOFTWARE

034 PROGRAMM ÜBERSICHT

037 ANLEITUNG 3D DRUCK

038 1. CAD ZU STL

044 2. STL ZU G-CODE

052 3. DRUCKER VORBEREITEN

054 4. DRUCKBETT-NIVELLIERUNG

056 5. HAFTUNG

060 6. DRUCKBETT VORBEREITEN

062 7. STARTE DRUCK

064 8. NACHBEARBEITUNG

067 SLICER PARAMETER

086 ABBILDUNGSVERZEICHNIS

092 LITERATURVERZEICHNIS

INTRODUCTION

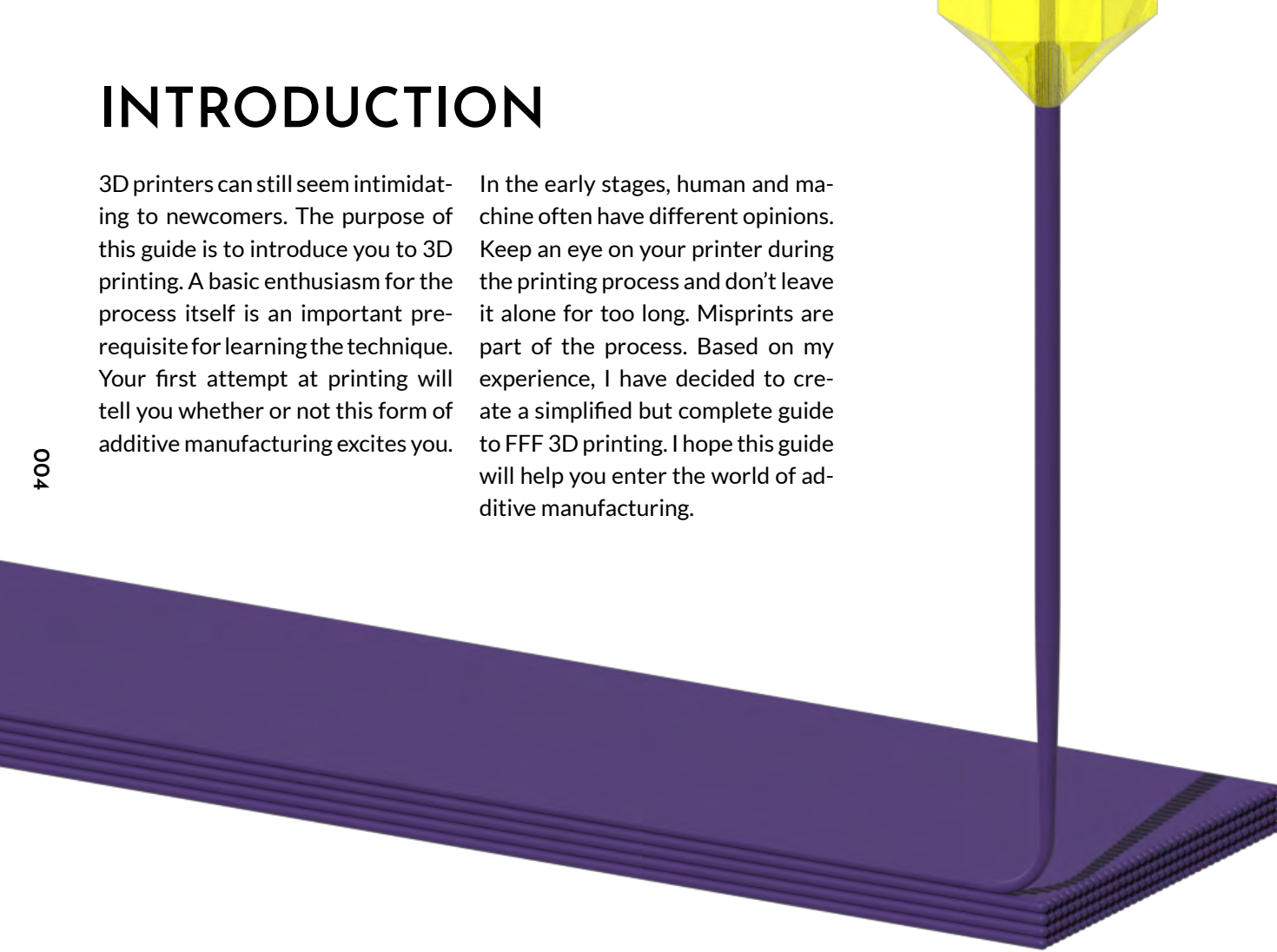
3D printers can still seem intimidating to newcomers. The purpose of this guide is to introduce you to 3D printing. A basic enthusiasm for the process itself is an important prerequisite for learning the technique. Your first attempt at printing will tell you whether or not this form of additive manufacturing excites you.

In the early stages, human and machine often have different opinions. Keep an eye on your printer during the printing process and don't leave it alone for too long. Misprints are part of the process. Based on my experience, I have decided to create a simplified but complete guide to FFF 3D printing. I hope this guide will help you enter the world of additive manufacturing.

EINFÜHRUNG

3D Drucker sind immer noch eine Herausforderung für viele Interessierte. Mit der folgenden Anleitung möchte ich Dir das 3D Drucken näher bringen. Eine Grundbegeisterung für den Prozess selbst ist eine wichtige Voraussetzung für das Erlernen des Verfahrens. Bei deinem ersten Druckversuch wirst Du herausfinden, ob dich diese Form der Additiven Fertigung begeistert oder nicht.

In der Anfangsphase kommt es häufig zu Meinungsverschiedenheiten zwischen Mensch und Maschine. Beobachte deinen Drucker während des Druckprozesses und lasse ihn nicht zu lange alleine. Fehldrucke gehören dazu. Aufgrund meiner Erfahrung habe ich beschlossen eine vereinfachte, aber vollständige Anleitung zum FFF 3D Druck zu erarbeiten. Ich hoffe, dass diese Einführung Dir den Zugang in die Welt der Additiven Fertigung erleichtert.



ADDITIVE MANUFACTURING

Three-dimensional printing describes a technique in which material is deposited layer by layer. In industry, this method is used for pre-production or production (Sitotaw et al., 2020). A major advantage of this method is the efficient use of material. In traditional subtractive manufacturing methods, material is removed to achieve the desired shape. This often leads to excessive material consumption (Berman, 2012).

Additive Manufacturing describes technologies that create physical objects through the continuous addition of mostly polymer-based material (Ligon et al., 2017).

Fused Filament Fabrication (FFF) also known as Filament Deposition Modeling (FDM) belongs to the additive manufacturing methods (Gebhardt, 2013). Other techniques such as stereolithography (SLA) (Grothe, Brockhagen and Storck, 2020), binder jetting (Gibson et al., 2021) or directed energy deposition (Svetlizky et al., 2021), (Gieseke et al., 2016) allow 3D printing with materials other than just polymers and are also categorized as Additive Manufacturing (Sitotaw et al., 2020).

This guide only focuses on the additive manufacturing method Fused Filament Fabrication. If you want to try another 3D printing technique besides FFF, stereolithography is often a good choice, as Resin 3D Printers, e.g. from *Formlabs GmbH*, are commercially available and you do not need access to large industrial machines.

ADDITIVE FERTIGUNGSVERFAHREN

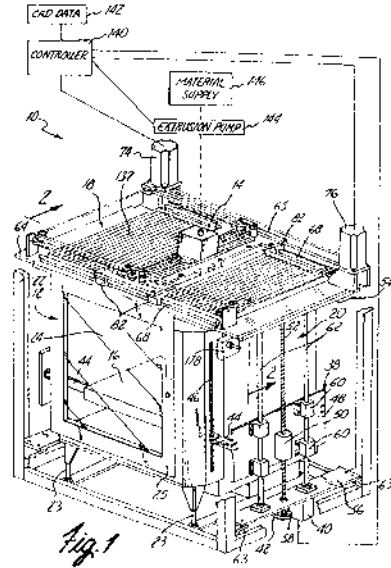
Der dreidimensionale Druck beschreibt eine Technik, bei der das Material Schicht für Schicht aufgebracht wird. In der Industrie wird diese Methode für die Vorproduktion oder die Produktion eingesetzt (Sitotaw et al., 2020). Ein großer Vorteil bei dieser Methode ist die effiziente Materialnutzung. Bei herkömmlichen subtraktiven Fertigungsverfahren wird dagegen Material abgetragen, um die gewünschte Form zu erhalten. Das führt oft zu einem übermäßigen Materialverbrauch (Berman, 2012).

Additive Fertigung beschreibt Technologien, welche physikalische Objekte durch die kontinuierliche Zugabe von meist polymerbasiertem Material schaffen (Ligon et al., 2017).

Fused Filament Fabrication (FFF) auch bekannt als Filament Deposition Modeling (FDM) gehört zu den additiven Fertigungsverfahren (Gebhardt, 2013). Andere Techniken wie Stereolithografie (SLA) (Grothe, Brockhagen and Storck, 2020), Binder Jetting (Gibson et al., 2021) oder directed energy deposition (Svetlizky et al., 2021), (Gieseke et al., 2016) ermöglichen das 3D Drucken mit anderen Materialien als nur Polymeren und werden ebenfalls zur Additiven Fertigung gezählt (Sitotaw et al., 2020).

Diese Anleitung befasst sich ausschließlich mit dem Additiven Fertigungsverfahren Fused Filament Fabrication (FFF). Wenn du neben der FFF Methode eine andere 3D Drucktechnik ausprobieren möchtest, eignet sich oft die Stereolithografie, da hier auch kommerzielle Resin 3D Drucker z.B. von der *Formlabs GmbH* im Handel erhältlich sind und man keinen Zugang zu großen industriellen Maschinen benötigt.

3D PRINTER MANUFACTURER



technical drawing from Patent US6722872B1/
technische Zeichnung aus Patent US6722872B1

The FFF method is also known by the acronym FDM, which stands for Fused Deposition Modeling.

Basically, it is the same process. The two different terms date back to the 1990s, when *Stratasys Inc* patented the FDM process (Swanson et al., 2004). The term FDM was trademarked and we now refer to the process as FFF printing. *Stratasys Inc* is still one of the technology leaders in additive manufacturing.

UltiMaker is another successful company that was founded in the Netherlands in 2011 (Chapman, 2022). *UltiMaker* developed the slicing application *UltiMaker Cura*, which is used worldwide, mainly in the private sector (*UltiMaker*, 2011). An *UltiMaker* printer is very user-friendly, but also very expensive.

On the other hand, the Chinese company *Shenzhen Creality 3D Technology Co., Ltd*, also known as *Creality*, produces less expensive printers (*creality*, 2022).



As the name suggests, the Berlin-based start-up *BigRep GmbH* has further developed large-format 3D printing (*BigRep*, 2018). This scaling of conventional desktop 3D printers enables the direct production of furniture or even vehicle bodies.

3D DRUCKER HERSTELLER

Das FFF-Verfahren ist auch bekannt unter der Abkürzung FDM, welche für Fused Deposition Modeling steht. Im Prinzip handelt es sich um das gleiche Verfahren. Die unterschiedlichen Bezeichnungen gehen auf die 90er Jahre zurück, in welchen die Firma *Stratasys Inc* Patente auf das FDM-Verfahren angemeldet hat (Swanson et al., 2004).



Aufgrund dessen ist die Formulierung FDM geschützt und wir nennen das Verfahren nun FFF-Verfahren. *Stratasys Inc* gehört auch heute noch zu den weltweit führenden Unternehmen im Bereich der Additiven Fertigung.



UltiMaker

UltiMaker ist ein anderes erfolgreiches Unternehmen, welches 2011 in den Niederlanden gegründet wurde (Chapman, 2022). *UltiMaker* entwickelte die Slicing Software *UltiMaker Cura*, welche weltweit am häufigsten im privaten Bereich Anwendung findet (*UltiMaker*, 2011). Ein Drucker von *UltiMaker* ist sehr nutzerfreundlich, jedoch auch sehr hochpreisig.

Im Gegensatz dazu stellt das chinesische Unternehmen *Shenzhen Creality 3D Technology Co., Ltd.*, auch bekannt als *Creality*, kostengünstigere Drucker her (*creality*, 2022).



Das Start-up *BigRep GmbH* aus Berlin hat, wie der Name schon sagt, den großformatigen 3D Druck weiterentwickelt (*BigRep*, 2018). Diese Skalierung herkömmlicher Desktop-3D-Drucker ermöglicht die direkte Herstellung von Möbeln oder sogar Fahrzeugkarosserien.

3D PRINTER MANUFACTURER



carbon fiber reinforced 3D printed parts/
kohlefaserverstärkte 3D-Druckteile

To reinforce 3D printed components, with a continuous filament containing carbon fibres, the company Mark3D GmbH has developed the Markforged Mark Two 3D printer (Mark3D, 2016).



Machines from Prusa Research are also popular for experimental 3D printing, as they are easy to modify. The Czech company was founded in 2012 by Josef Prusa, who initially pursued 3D printing as a hobby. Prusa's printers offer good value for money (Prusa, 2022).

The latest generation of desktop 3D printers are faster, easier to maintain and supported by artificial intelligence. *The Bambu Lab P1P* or the *Crealty K1* are advanced Desktop 3D printers (Bambu Lab, 2023). traditional 3D printing simpler and more accessible (Bambu Lab, 2023).

PRUSA
RESEARCH
by **JOSEF PRUSA**

3D DRUCKER HERSTELLER



example for 3D printed furniture/
Beispiel für 3D-gedruckte Möbel,
Bow chair (left/ links) & Rise Chair (right/rechts)
by Nagami x Zaha Hadid Architects

Um 3D gedruckte Bauteile mit einem durchgehenden Filament, welches Kohlefasern enthält, zu verstärken, hat die Firma *Mark3D GmbH* den 3D Drucker *Markforged Mark Two* entwickelt (Mark3D,2016).

Sehr beliebt für den experimentellen 3D Druck sind auch die Geräte von *Prusa Research*, da sie sich leicht modifizieren lassen. Das tschechische Unternehmen wurde 2012 von Josef Prusa ins Leben gerufen, welcher das 3D Drucken anfangs nur als Hobby betrieb. Die Drucker von *Prusa* haben ein sehr gutes Preis-Leistungs-Verhältnis (Prusa, 2022).

Die neueste Generation von Desktop 3D Druckern sind schneller, pflegeleichter und werden durch Künstliche Intelligenz unterstützt. Der *Bambu Lab P1P* oder der *Crealty K1* sind führende Geräte der weiter entwickelten Technologie für den privaten Haushalt (Bambu Lab, 2023).



3D PRINTER

In addition to the manufacturers presented here, there are of course many other companies dedicating their efforts to 3D printing.

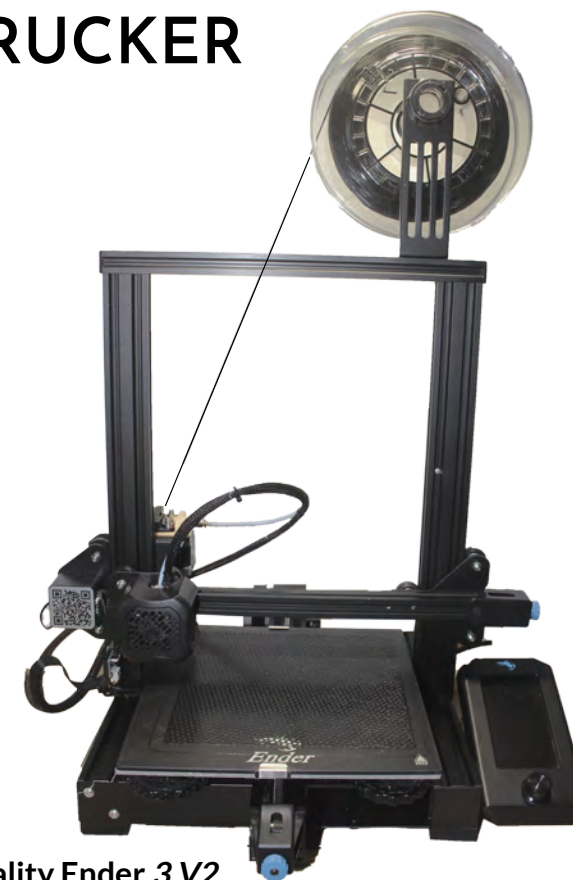
The companies mentioned have especially come across my 3D printing journey. In this guide we will work with the *Prusa i3 MK3S+* and in the 4D printing Guide we will work with the *Creality Ender 3 V2*. Once you have understood the principle of 3D printing, the knowledge can be applied to printers from different manufacturers, since the machines mostly are assembled of recurring components.



Prusa i3 MK3S+

3D DRUCKER

Neben den vorgestellten Herstellern gibt es natürlich noch viele weitere Firmen, welche sich dem 3D Druck widmen. Die hier genannten Firmen sind vor allem mir persönlich beim 3D Drucken über den Weg gelaufen. In dieser Anleitung wird mit dem *Prusa i3 MK3S+* gearbeitet und in der 4D Druck Anleitung mit dem *Creality Ender 3 V2*. Wenn man das Prinzip des 3D Druckens einmal verstanden hat, kann das Wissen auf Drucker verschiedener Hersteller angewandt werden, da die Maschinen mit immer wiederkehrenden Bauteilen bestückt sind.

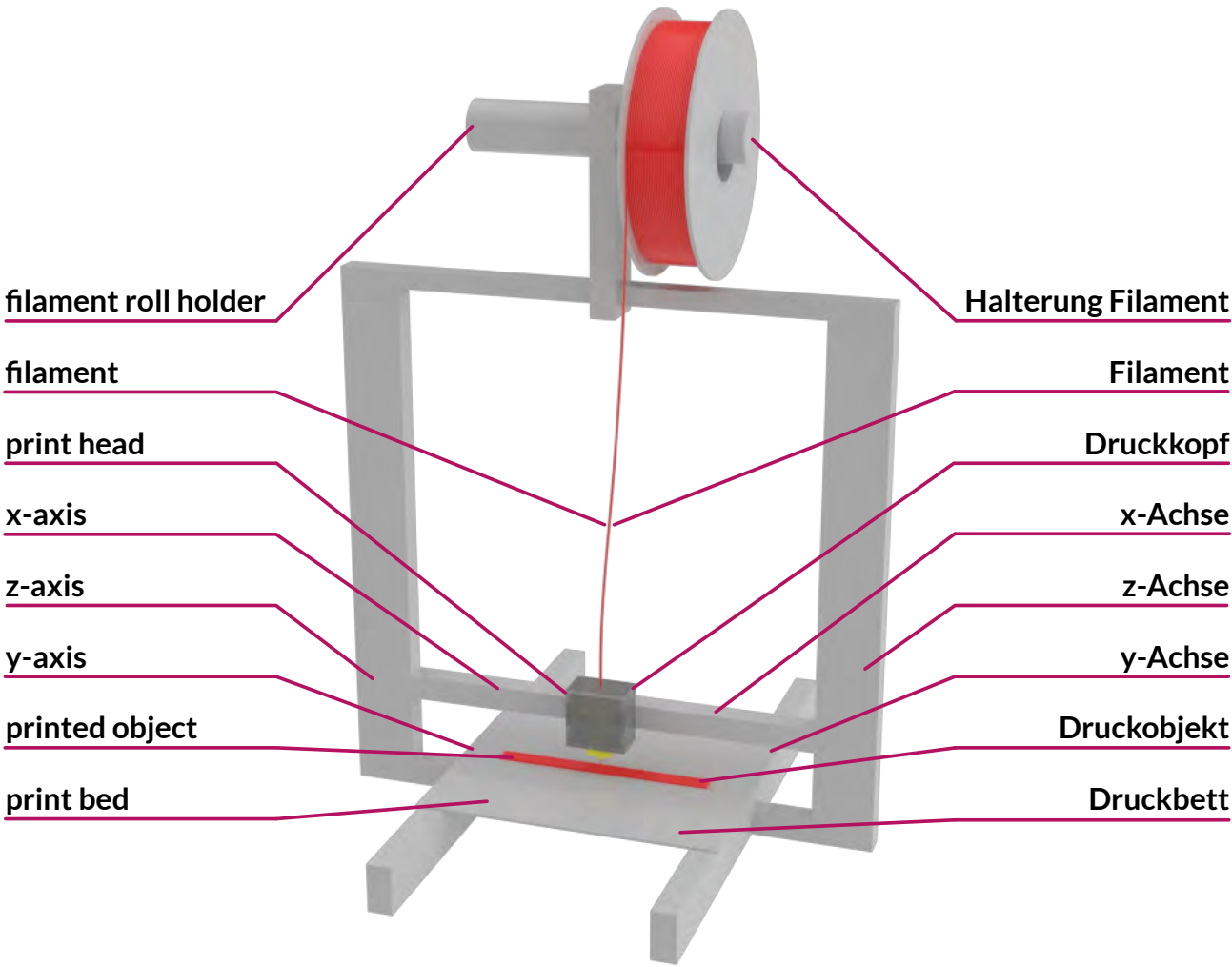
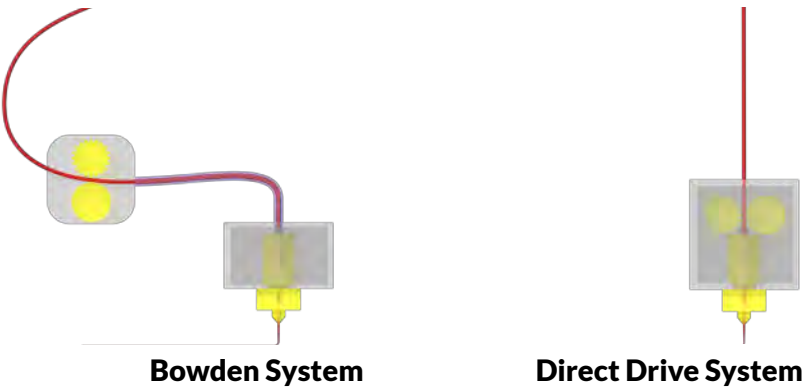


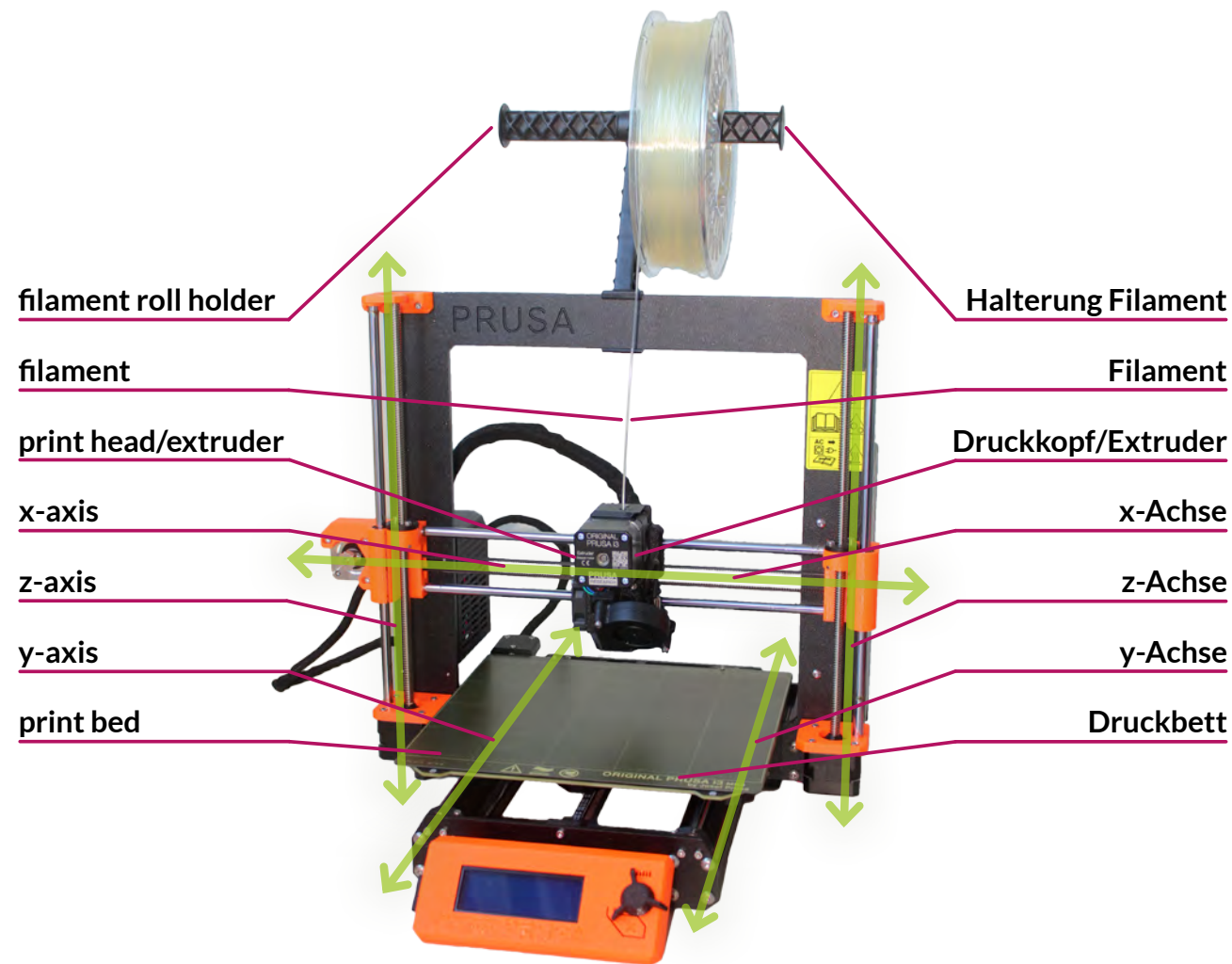
Creality Ender 3 V2

STRUCTURE 3D PRINTER

The abstract illustrations on the following pages are based on the 3D printer model *Prusa i3 MK3S+*. The **print head** moves in the FFF method on two of the three axes of a cartesian coordinate system. On the *Prusa i3 MK3S+*, the printhead operates on the **x-axis** and **z-axis**. The movement along the **y-axis**, which is required for the entire three-dimensional printing process, gets executed by the **print bed**. Print beds on other conventional printers, such as the *Ultimaker3 extended*, travel on the z-axis. The print head in these printers changes their position on the x-axis and y-axis (Dönitz et al., 2023). The **filament** is guided from the **filament roll holder** to the transport rollers in the feeder. Note

that there are two different feeding mechanisms. A printer with a Direct Drive system has transport rollers directly above the heat sink in the print head. A printer with a Bowden system has a feeder close to the filament roll holder. The filament is fed through a Teflon tube to the heat sink in the print head. With the Bowden system, The heavy stepper motor that drives the transport rollers is not located directly on the printhead, allowing faster and more precise printhead movements. But this system complicates the printing of flexible filaments (Ahlers, 2018). The model shown here has a **Direct Drive** system. The printer used for 4D printing, *Creality Ender 3 V2*, has a **Bowden** system.



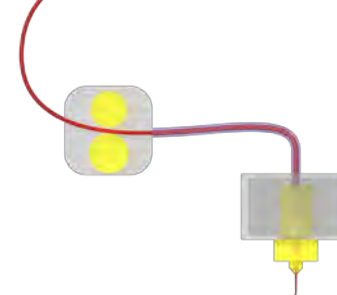


AUFBAU 3D DRUCKER

Die abstrahierten Veranschaulichungen auf den folgenden Seiten basieren auf dem 3D Druckermmodell *Prusa i3 MK3S+*. Bei der FFF-Methode bewegt sich der **Druckkopf** auf zwei der drei Achsen des dreidimensionalen Koordinatensystems. Bei dem *Prusa i3 MK3S+* bewegt sich der Druckkopf entlang der **x-Achse** und **z-Achse**. Die Bewegung auf der dritten Achse, in diesem Fall der **y-Achse**, wird vom **Druckbett** übernommen und vervollständigt somit den gesamten dreidimensionalen Druckprozess. Andere herkömmliche Drucker haben Druckbetten, die sich auf der z-Achse bewegen, wie der *Ultimaker3 extended*. Daraus resultierend übernimmt der **Druckkopf** bei diesen Druck-



Direct Drive System



Bowden System

ern den Positionswechsel auf der x- und y-Achse (Dönitz et al., 2023). Das **Filament** wird von der **Halterung für Filamentrollen** aus zu den Transportrollen im Feeder geführt. Es wird dabei zwischen zwei verschiedenen Feeding Mechanismen unterschieden. Bei einem Drucker mit Direct Drive System befinden sich die Transportrollen direkt über dem Kühlkörper im Druckkopf. Bei einem Drucker mit Bowden System liegt der Feeder in der Nähe von der Halterung für Filamentrollen. Das Filament wird durch einen Teflonschlauch hin zum Kühlkörper

er in den Druckkopf geführt. Der schwere Schrittmotor, welcher die Transportrollen antreibt, befindet sich beim Bowden System nicht direkt am Druckkopf und ermöglicht somit schnellere und präzisere Bewegungen des Druckkopfes. Allerdings erschwert dieses System das Drucken von flexiblen Filamenten (Ahlers, 2018). Das hier abgebildete Modell verfügt über ein '**Direct Drive**' System. Der für den 4D Druck verwendete Drucker, *Crealiti Ender 3 V2*, verfügt über ein '**Bowden**' System.

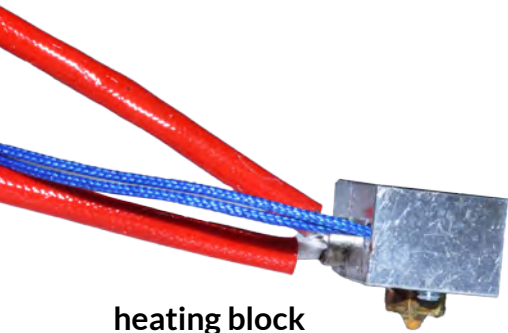
PRINT HEAD - EXTRUDER



heat break



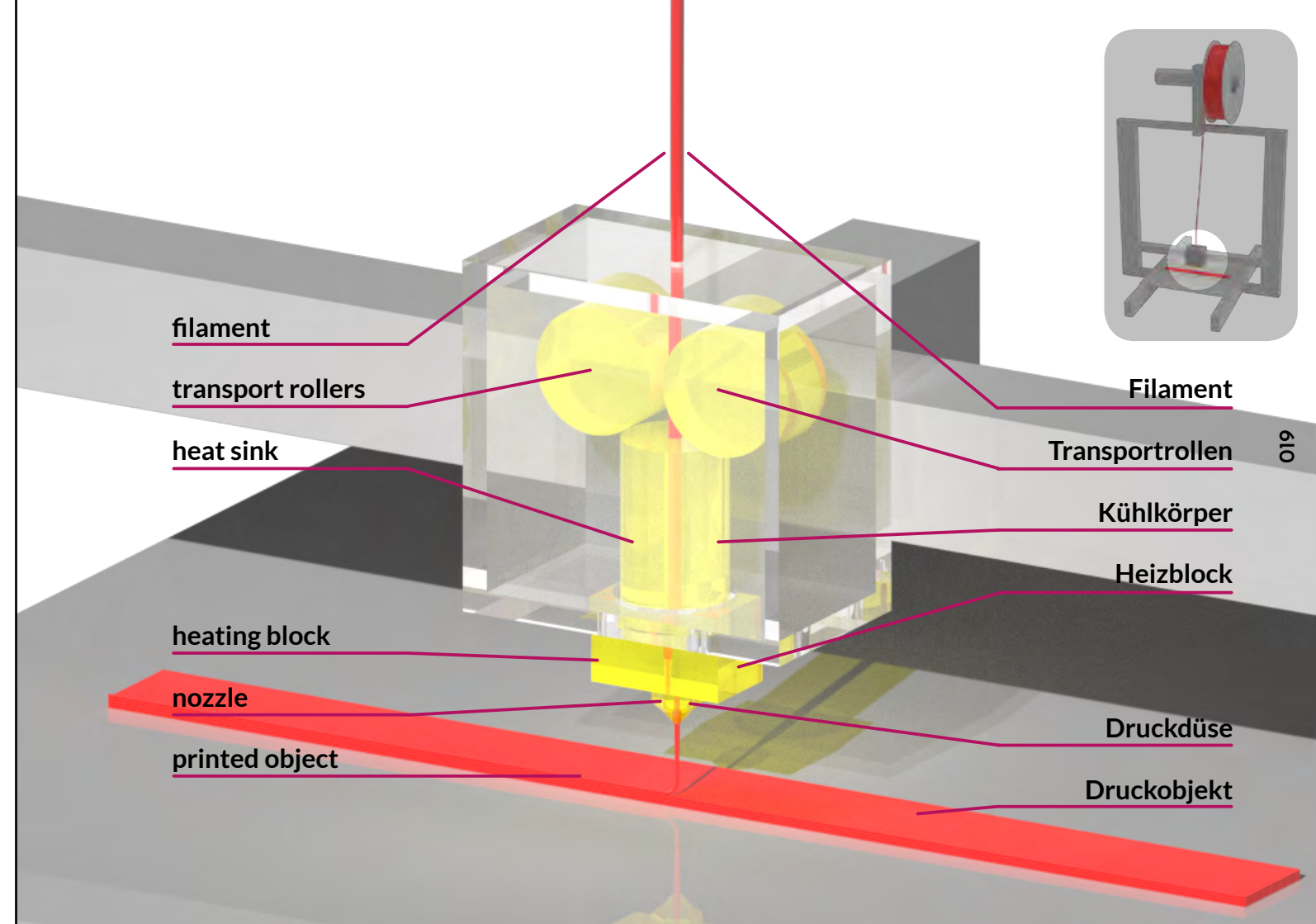
heat sink



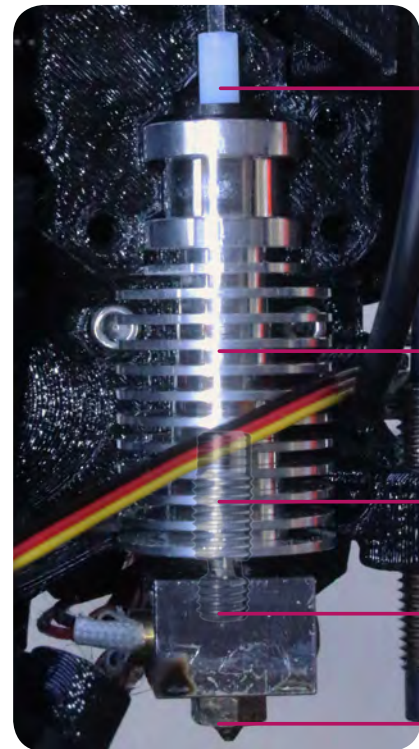
heating block

The **filament** is fed into the print head through **transport rollers**, which then undergoes heating in the **heating block** (Dönitz et al., 2023). The **heat sink** is usually made of metal and prevents the **extruder** from overheating. Due to its geometry, it can distribute the heat and thus prevent a clogging of the **nozzle**. The cold heat sink is connected to the hot heating block by a **heat break**, which controls the melting process of the filament. The heat break is usually made of stainless steel with a titanium alloy or is

available as a bi-metal heat break with a copper alloy on the cold side to ensure a more precise temperature transition. In the heating block, the nozzle is directly connected to the heat break. Two cables are attached to the Heating block. The heating cartridge (red) heats the heating block and the thermistor (blue) measures and communicates the temperature to the printer display.



Prusa i3 MK3S+



Teflon Tube / Teflonschlauch

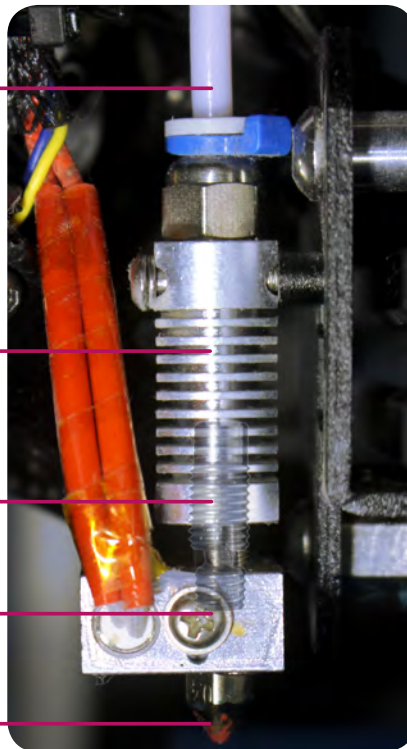
heat sink / Kühlkörper

heat break / Halsdüse

heating block / Heizblock

nozzle / Druckdüse

Crealty Ender 3 V2

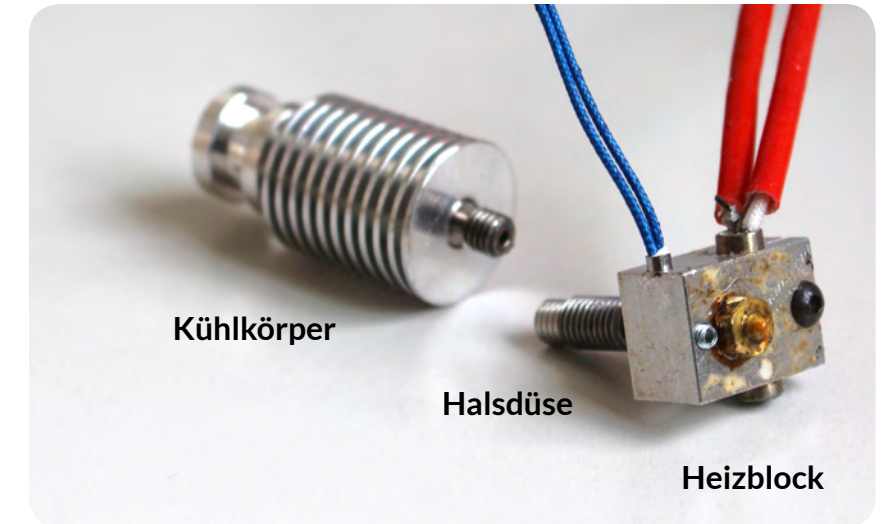


DRUCKKOPF - EXTRUDER

Das Filament wird über Transportrollen in den Druckkopf geführt und anschließend im Heizblock erhitzt (Dönitz et al., 2023). Das heat sink ist meist aus Metall und fungiert als Kühlkörper, um eine Überhitzung im Extruder zu vermeiden. Es kann aufgrund seiner Geometrie die Wärme verteilen und somit Verstopfungen in der Druckdüse vorbeugen. Der Kühlkörper, die kalte Hälfte des Extruders, ist mit dem Heizblock, der heißen Hälfte des Extruders, durch eine Halsdüse verbunden, welches den Schmelzvorgang des Filaments kontrolliert. Die Halsdüse besteht meist aus Edelstahl mit einer Titan-Legierung oder wird als Bi-metal-Halsdüse mit einer Kupferlegierung auf der kalten Seite versehen um

einen präziseren Temperaturübergang zu gewährleisten. Im Heizblock dockt die Druckdüse direkt an die Halsdüse. Am Heizblock sind zwei Kabel angeschlossen. Die Heizpatrone (rot) erhitzt den Heizblock

und der Temperaturfühler (blau) misst und kommuniziert die Temperatur an das Druckerdisplay.



Kühlkörper

Halsdüse

Heizblock

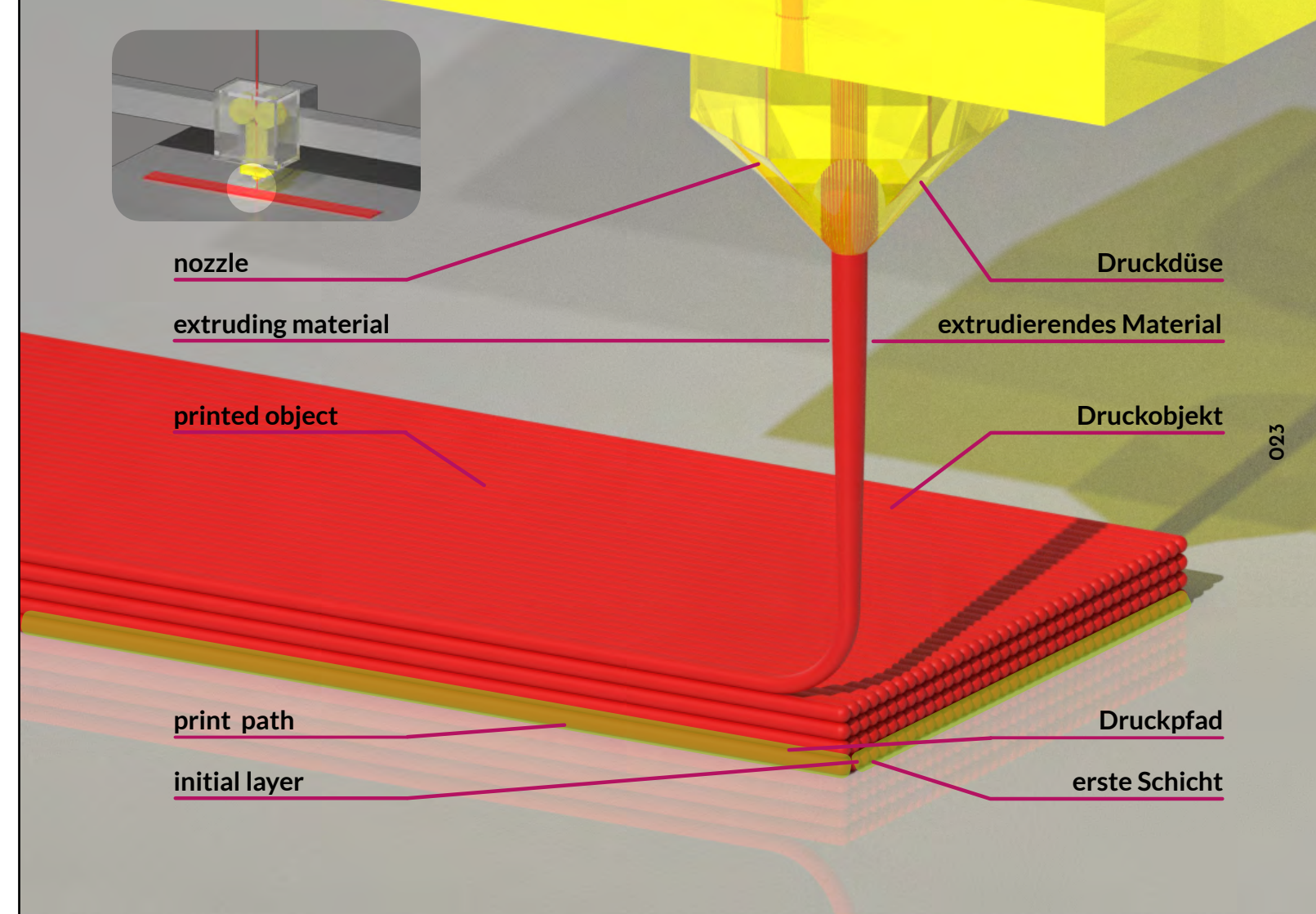
PRINTED OBJECT

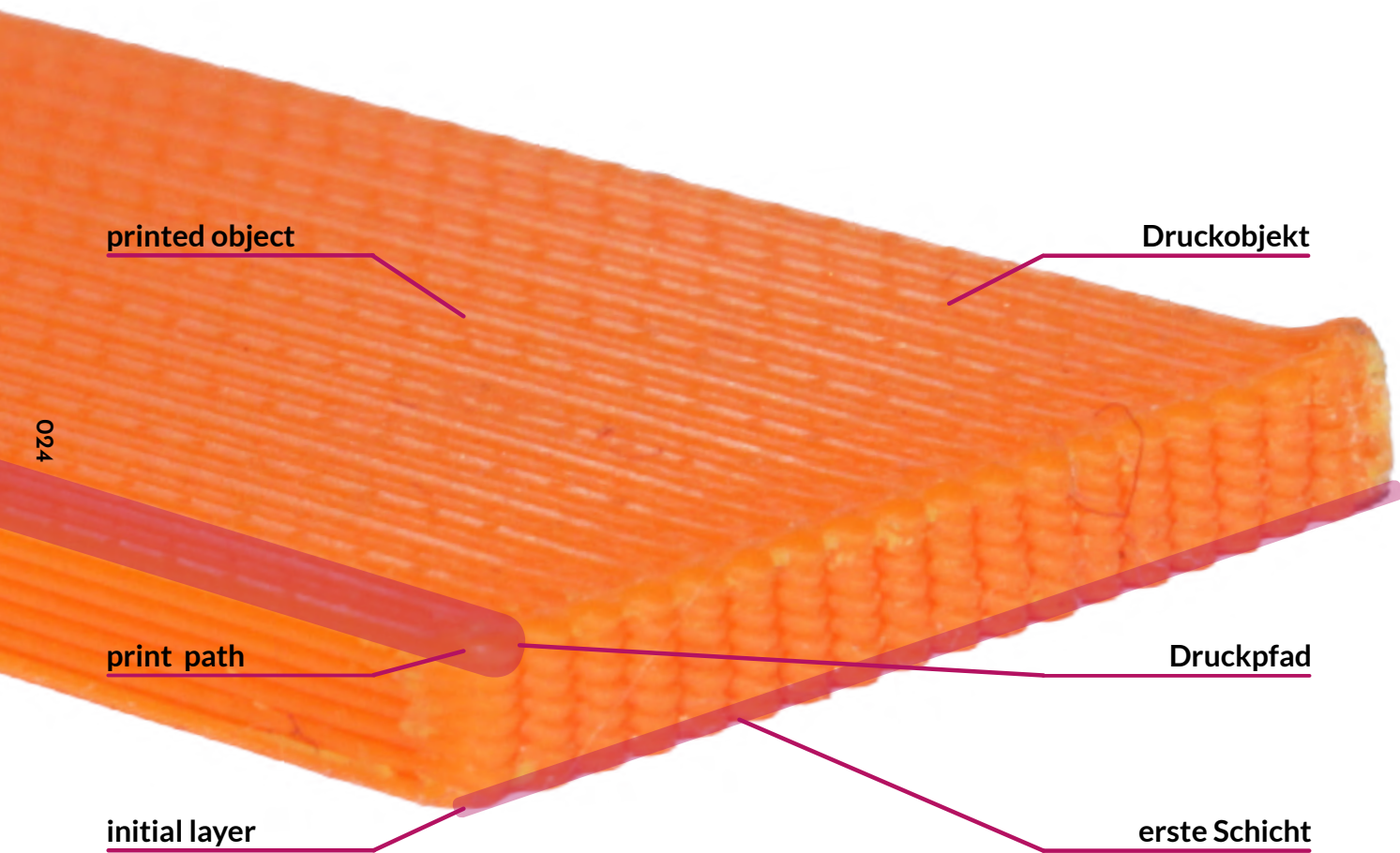


nozzle / Druckdüse

Once melted, the material is extruded through the **nozzle** and applied onto the print bed in a series of layers. One **layer** consists of several **print paths** (Dönitz et al., 2023). The sequence of print paths and the stacking of print layers form the 3D **printed object**.

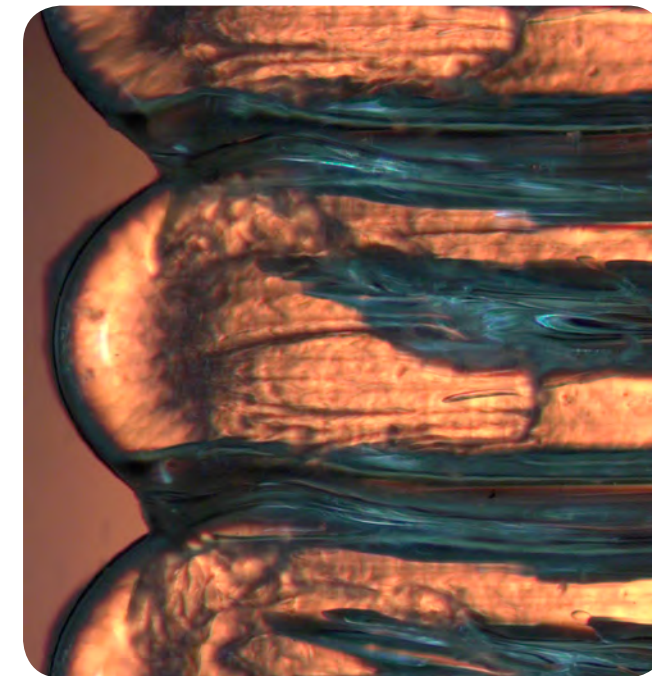
A print path can be defined in its width and height. Most 3D printers are equipped with a nozzle of 0.4mm diameter, so the Line Width is usually also 0.4mm. The parameters Line Width and Layer Height are defined together with other print settings in the Slicing Software.





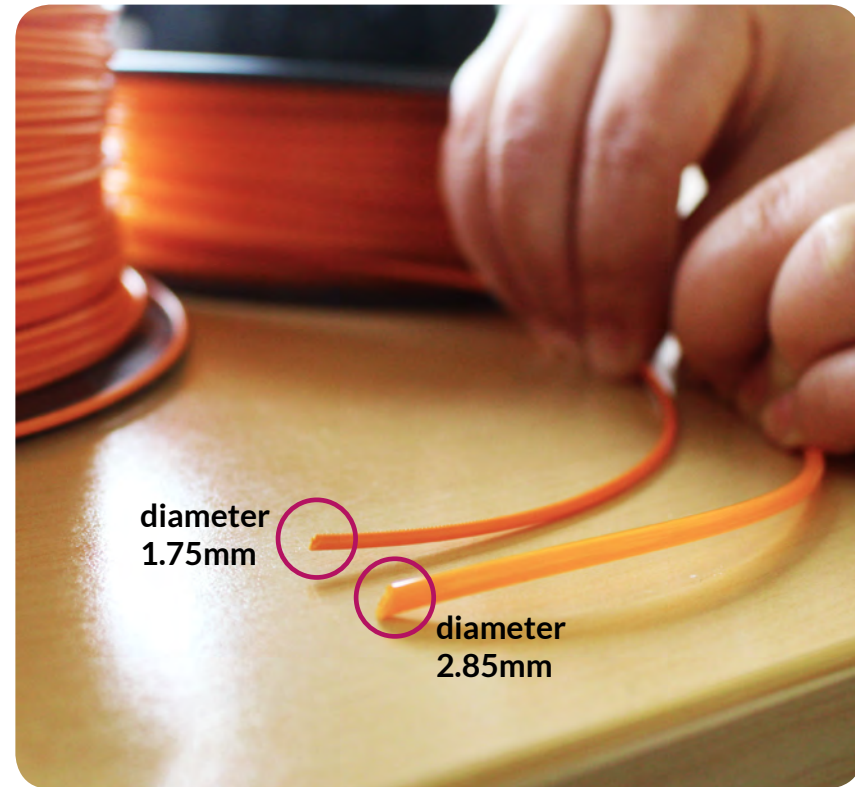
DRUCKOBJEKT

Sobald das **Material** geschmolzen ist, wird es durch die **Druckdüse** extrudiert und in mehreren Schichten auf das Druckbett aufgetragen. Eine **Schicht** besteht aus mehreren Druckpfaden (Dönitz et al., 2023). Die Aneinanderreihung von Druckpfaden und die Überlagerung von Druckschichten bilden dann das 3D **gedruckte Objekt**. Ein **Druckpfad** kann in seiner Breite und Höhe definiert werden. Die meisten 3D Drucker sind mit einer Düse von 0,4mm Durchmesser ausgestattet, weswegen die Linienbreite meist auch 0,4mm beträgt. Die Parameter Linienbreite und Schichtdicke legt man zusammen mit weiteren Druckeinstellungen im Slicer fest.



Microscope image of print paths (PLA) /
Mikroskopaufnahme von Druckpfaden (PLA)

MATERIAL FILAMENT



Commercially available filaments usually come in two diameters. Filaments with a diameter of 1.75mm are for example used in the *Prusa i3 MK3S+* and filaments with a diameter of 2.85mm in the *Ultimaker3 extended*. Usually smaller printers are provided with the thinner filament and larger printers with the thicker filament. The diameter of the filament only affects the printing time and not the quality of the final product. The most common polymer for 3D printing is polylactic acid (PLA). PLA is easy to handle, adheres well to the print bed and can be recycled industrially. Another commonly used polymer is acrylonitrile butadiene styrene (ABS), which is lighter and more im-

MATERIAL FILAMENT

Kommerziell verfügbare Filamente gibt es meist in zwei Durchmessern. Filamente mit einem Durchmesser von 1.75mm werden beispielsweise im *Prusa i3 MK3S+* eingesetzt und Filamente mit einem Durchmesser von 2.85mm im *Ultimaker3 extended*. Meist werden kleinere Drucker mit dem dünneren Filament ausgestattet und größere Drucker mit dem dickeren Filament. Der Durchmesser des Filaments wirkt sich nur auf die Druckzeit aus und nicht auf die Qualität des Endprodukts. Das gängigste Polymer zum 3D Druck ist Polymilchsäure, kurz PLA. PLA ist einfach zu handhaben, haftet gut auf dem Druckbett und kann industriell recycelt werden. Ein anderes häufig genutztes Polymer ist Acrylnitril-Butadien-Sty-

rol (ABS), welches leichter und stoßfester ist als PLA. Jedoch setzt ABS beim Schmelzvorgang gesundheitsschädliche Dämpfe frei und ist weniger nachhaltig als PLA (Stephens et al., 2013). Wenn man ein weniger steifes Material benötigt, kommt oft thermoplastisches Polyurethan (TPU) zum Einsatz. TPU gibt es in verschiedenen Shore-Härten, welche die flexible Verformbarkeit des Materials einordnen. TPU bewegt sich zwischen den Härtegraden 60A – 90A auf der Shore-Härteskala. Wobei 60A circa der Härte eines Fahrradreifens entspricht und 90A vergleichbar ist mit den Rollen eines Einkaufswagens. Allerdings kann es tückisch sein sehr weiches TPU zu drucken, da es sich gerne in den Transportrollen

des Feeders verfängt. Es gibt natürlich auch sehr spezielle Filamente auf dem Markt wie das kompostierbare *WillowFlex* von der *3dk Trading GmbH* oder Filamente, welche zu einem gewissen Anteil mit Holz- oder Hanffasern verstärkt werden. Auch Filamente die Metallpartikel wie Kupfer enthalten oder leitfähige Filamente auf Kohlenstoffbasis sind im Handel erhältlich. In der Luft- und Raumfahrt werden sogar Materialien eingesetzt, welche einen Formgedächtniseffekt haben (Aerospace 3D printing | Orion Additive Manufacturing). Oft kann man Firmen auch kontaktieren, da diese oft Reste oder alte Filamente haben, welche Sie gerne an Studierende verschenken. Beachte dabei, dass alte Filamente, die

MATERIAL FILAMENT

compact resistant than PLA. But one should consider that ABS releases unhealthy gases during the melting process and is less sustainable than PLA (Stephens et al., 2013). When you need a less rigid material, thermoplastic polyurethane (TPU) is often used. TPU comes in different Shore hardness, which rank the flexible formability of the material. TPU usually ranges from 60A to 90A on the Shore hardness scale. Whereby 60A corresponds approximately to the hardness of a bicycle tire and 90A is comparable to the rollers of a shopping cart. Printing very soft TPU can be tricky, as it tends to get caught in the transport rollers of the feeder. The market also offers very special filaments,

such as the compostable filament *WillowFlex* from *3dk Trading GmbH* or filaments that are reinforced to a certain percentage with wood or hemp fibers. Filaments that contain metal particles such as copper or conductive carbon-based filaments are also commercially available. In the aerospace industry, even materials are used that have a shape memory effect (Aerospace 3D printing | Orion Additive Manufacturing). One can also try to contact companies, as they often have leftovers or old filaments, which they are happy to give away to students. Please note that old filaments that have not been stored in a vacuum should get dried first. Berlin-based distributors for fila-

ments are for example *Filafarm GmbH & Co. KG* or *3dk Trading GmbH*.

MATERIAL FILAMENT

nicht vakuumiert gelagert wurden, vorerst getrocknet werden sollten. In Berlin ansässige Vertriebe für Filamente sind beispielsweise die *Filafarm GmbH & Co. KG* oder die *3dk Trading GmbH*.



CAD MODELING SOFTWARE

CAD modeling softwares are used to create digital 3D models, which can be fabricated as physical objects through FFF 3D printing. The 3D model has to be exported as a STL format for 3D printing. When converted to an STL, the shell of the 3D object gets divided into triangles and a mesh results, which can then be further processed by a slicing software.

Blender and *FreeCAD* are open source CAD applications (Fastermann, 2016). *SketchUp Make* has a webbased free version where no program needs to be installed, only a login is required. For most paid programs, such as *Rhinoceros 3D* or *Fusion 360*, you can try a demo version for about three months or check for student discounts.



CAD MODELLIERUNGSSOFTWARE

Eine CAD Modellierungssoftware ist dazu da digitale 3D Modelle zu erstellen, die durch den FFF 3D Druck als physische Objekte gefertigt werden können. Für den 3D Druck wird das 3D Modell als ein STL-Format exportiert. Bei der Umwandlung in eine STL-Schnittstelle wird die Hülle des 3D Objektes in Dreiecke unterteilt und es entsteht ein Gitternetz, welches anschließend von einer Slicing Software weiter verarbeitet werden kann.

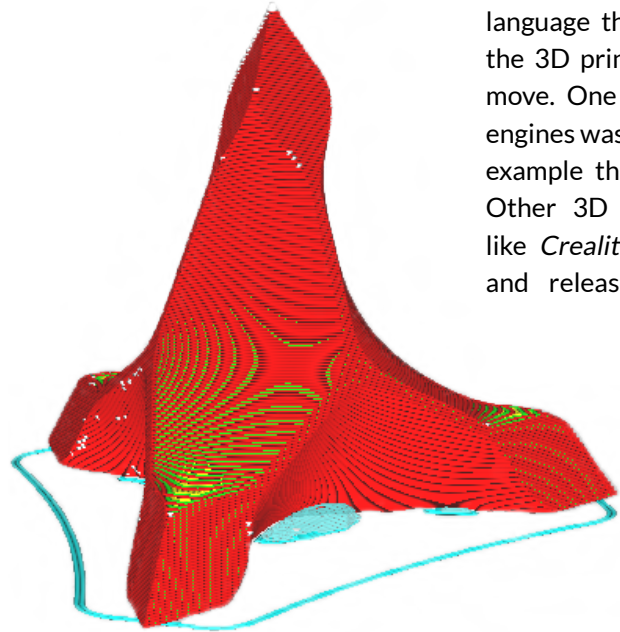
Blender und *Free-CAD* sind frei zugängliche CAD-Anwendungen (Fastermann, 2016). Bei *SketchUp Make* gibt es eine webbasierte kostenfreie Version, bei welcher man kein Programm installieren muss sondern sich nur anmelden braucht. Bei den meisten kostenpflichtigen Programmen wie *Rhinoceros 3D* oder *Fusion 360* kann man circa drei Monate lang eine Demoversion testen oder Ausschau nach Rabatten für Studierende halten.



SLICING APPLICATION

A slicing software initially assigns defined printing parameters to the triangular mesh (STL) and creates a so-called G-code (Ariffin et al., 2018). The G-code is a programming language that tells the motors on the 3D printer how and where to move. One of the first 3D slicing engines was the *Slic3r*, on which for example the *PrusaSlicer* is based. Other 3D printer manufacturers like *Creality* have also developed and released their own slicers.

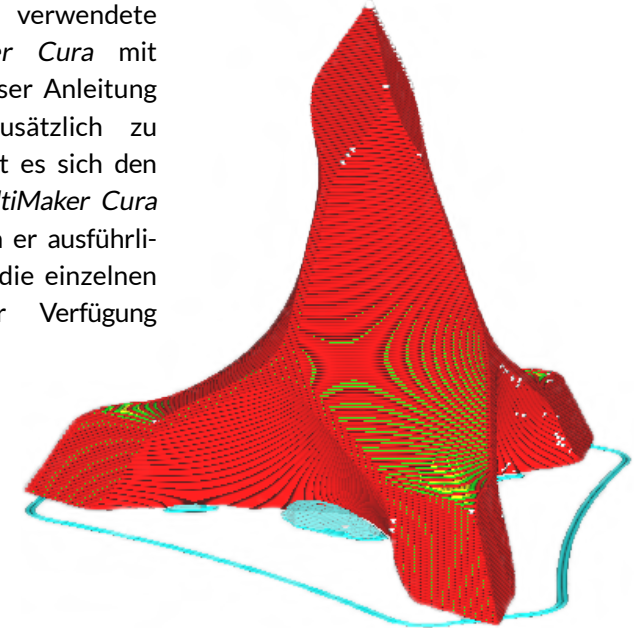
Most slicers are freely available. Of course, there are slicers like the *Simplify3D* software that can be purchased as well. In the end, it's like the 3D printers themselves and the CAD modeling software, as the slicers are very similar in their handling. The most commonly used slicer is *UltiMaker Cura*, which will also be used in this guide. In addition to *UltiMaker Cura*, it is worth downloading the Settings Guide for *UltiMaker Cura*, as it provides detailed explanations for the individual print parameters.



SLICING-SOFTWARE

Eine Slicing Software ist zunächst dafür da, dem kreierten Gitternetz (STL) definierte Druckparameter zuzuweisen und einen sogenannten G-Code zu erstellen (Ariffin et al., 2018). Der G-Code ist eine Programmiersprache, welche den Motoren am 3D Drucker befiehlt wie und wohin Sie sich bewegen sollen. Unter einen der ersten 3D-Slicing-Engines gab es *Slic3r*, auf welchem auch der *PrusaSlicer* basiert. Auch andere 3D Drucker Hersteller wie *Creality* haben Ihre eigenen Slicer entwickelt und heraus gebracht. Die meisten Slicer sind frei zugänglich. Es gibt natürlich auch Slicer wie die Software *Simplify3D*, welche man erwerben kann.

Am Ende ist es wie bei den 3D Druckern selbst und den CAD-Modellierungssoftwares, da die Slicer sich in ihrer Handhabung untereinander sehr stark ähneln. Der am häufigsten verwendete Slicer ist *UltiMaker Cura* mit welchem auch in dieser Anleitung gearbeitet wird. Zusätzlich zu *UltiMaker Cura* lohnt es sich den Settings Guide für *UltiMaker Cura* herunter zu laden, da er ausführliche Erklärungen für die einzelnen Druckparameter zur Verfügung stellt.



CAD PROGRAMMS



free version:
<https://www.blender.org/download/>



free version:
<https://www.freecad.org/downloads.php?lang=en>



free webbased version:
<https://www.sketchup.com/de/plans-and-pricing/compare>



90d test version:
<https://www.rhino3d.com/de/download/rhino/evaluation/>



30d test version:
<https://www.autodesk.de/products/fusion-360/free-trial>

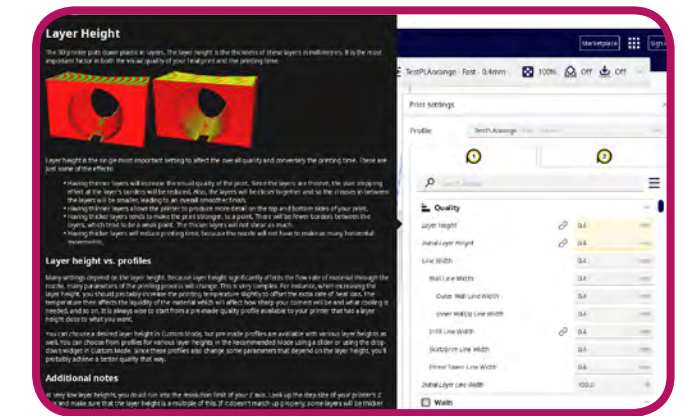
SLICERS



CURA download:
<https://ultimaker.com/de/software/ultimaker-cura>



CURA Settings Guide download:
<https://marketplace.ultimaker.com/app/cura/plugins/Ghostkeeper/SettingsGuide2>



3D PRINTING

A usually polymer-based material undergoes the melting process and gets extruded through a nozzle onto a printing bed. An object takes its three-dimensional shape layer by layer on the build plate. After the last layer the desired object can be removed from the build plate (Koch, Schmelzeisen and Gries, 2021). 3D printing allows us geometric freedom in designing and constructing lightweight structures (Dönitz et al., 2023).

We will place more emphasis on more sustainable production processes in future. FFF 3D printing offers an opportunity in this area, as the printing process uses only the amount of material the final printed product is made of. But to achieve this point, one or two failed prints end up in the trash as well. Furthermore, the biodegradability attributed to filaments such as polylactic acid (PLA) has to be taken with a pinch of salt. Green washing shouldn't be downplayed, but just criticizing won't get us more sustainable production processes either. We have to start researching somewhere, even if the start is sometimes a bit bumpy.



3D DRUCK

Ein meist polymerbasiertes Material wird geschmolzen und durch eine Düse auf eine Bauplatte extrudiert. Auf der Bauplatte nimmt ein Objekt Schicht für Schicht seine dreidimensionale Gestalt an. Anschließend wird es von der Bauplatte entfernt (Koch, Schmelzeisen und Gries, 2021). Der 3D Druck ermöglicht uns geometrische Freiheiten beim Gestalten und das Konstruieren leichter Strukturen (Dönitz et al., 2023). In der Zukunft werden wir mehr Wert legen auf nachhaltigere Produktionsprozesse. Der FFF 3D Druck bietet hier eine Option, da beim Druckprozess nur die Menge des Materials verwendet wird, aus welcher das gedruckte Endprodukt besteht.

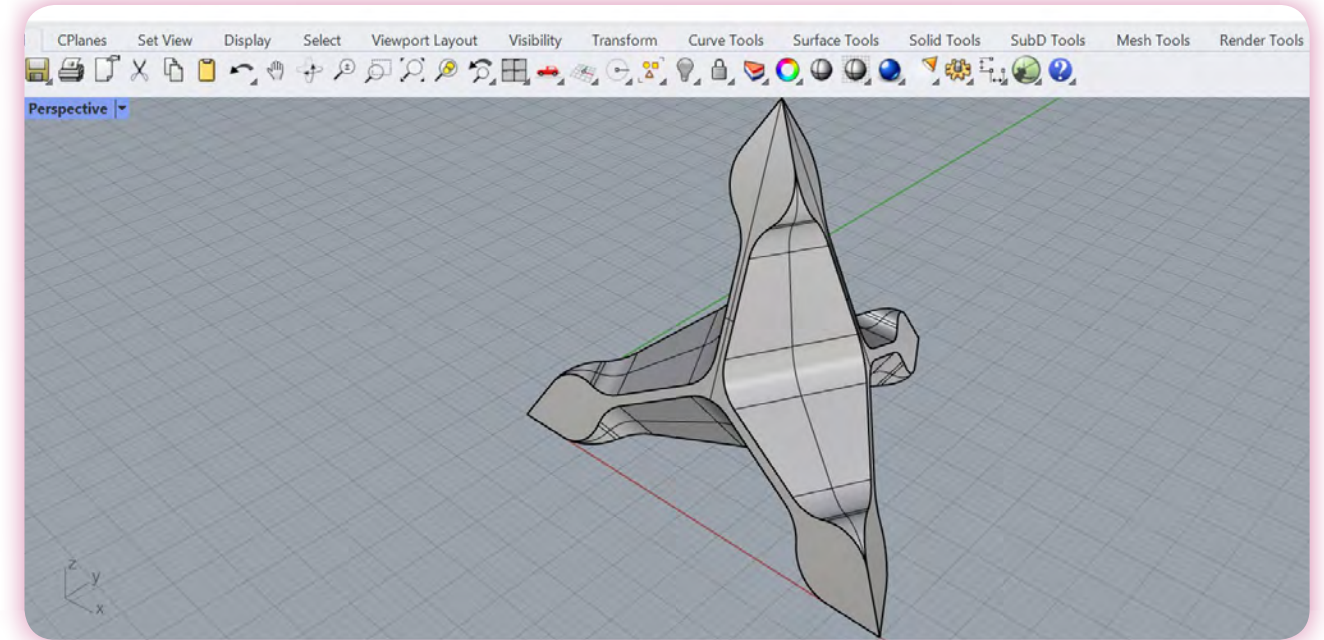
Um allerdings erstmal an diesen Punkt zu kommen, ladet auch der ein oder andere Fehldruck im Müll. Ebenso ist die biologische Abbaubarkeit, die Filamenten wie Polymilchsäure (PLA) zugesprochen wird, mit Vorsicht zu genießen. Green Washing sollte nicht verharmlost werden, aber nur Kritik zu üben bringt uns auch keine nachhaltigeren Produktionsprozesse. Irgendwo müssen wir anfangen zu forschen, auch wenn der Start manchmal etwas holprig verläuft.



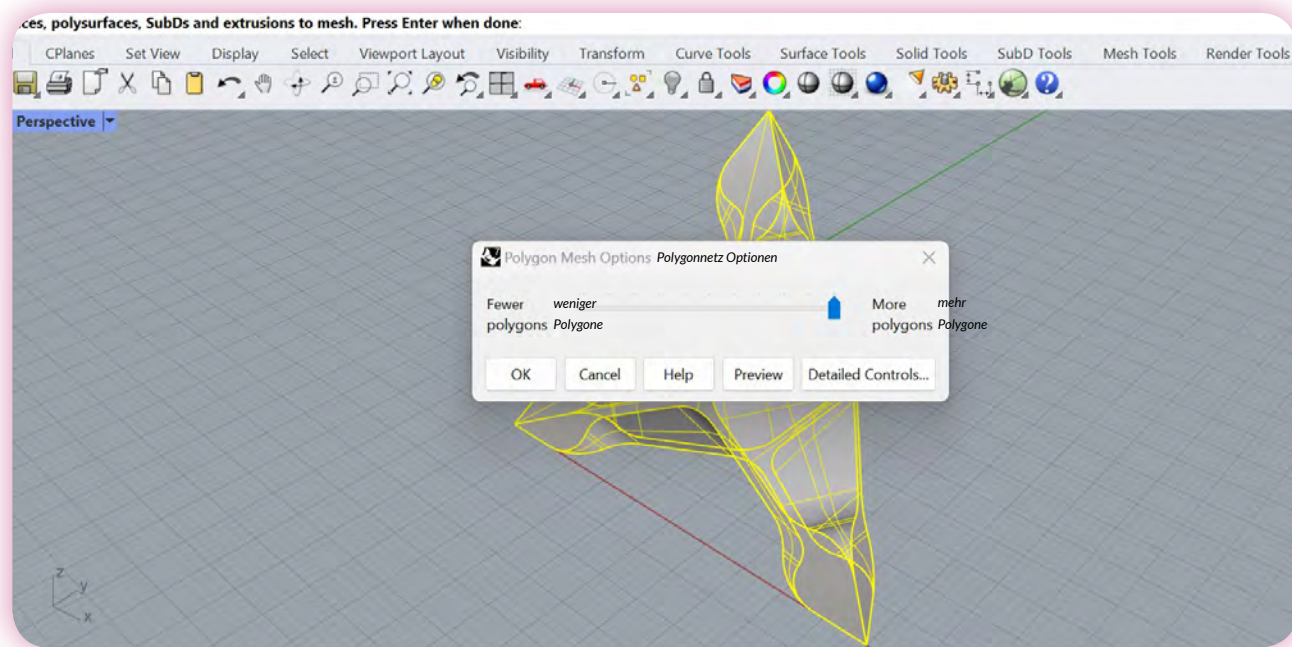
CAD MODEL TO STL

The desired 3D object can be modeled in a CAD program like *Rhinoceros 3D* or *FreeCAD*. In the following illustrations, the 3D object is processed with Rhinoceros 3D. This guide starts with a fully modeled CAD model. If you create the model yourself, it is usually a closed polysurface or a lightweight extrusion. The difference between a polysurface and an extrusion only becomes noticeable when the amount of data exceeds a certain size. Large accumulations of polysurfaces can become very sluggish and eat up a lot of memory, whereas extrusions are responsive and use little memory (McNeel Wiki - Lightweight Extrusion Objects, 2020). For further processing in the slicer, we need the STL file format respectively a

polygon mesh. The polysurface or extrusion can be converted into a polygon mesh directly in the CAD program or by exporting it to the STL format. When using the CAD software to convert the model into a mesh, you can usually define the number of polygons that are responsible for the level of detail. 3D models that can be downloaded from websites are usually already a finished polygon mesh.



3D Model from Polysurface
3D Modell aus Flächenverband

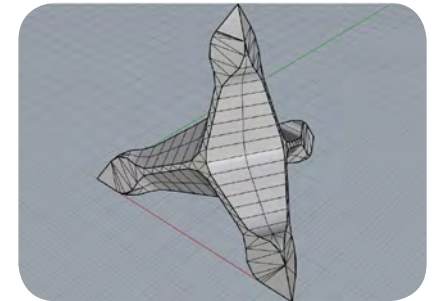


command: mesh -> window opens: choose amount of polygons -> OK
 Befehl: Polygonnetz -> Fenster öffnet sich: Anzahl der Polygone wählen -> OK

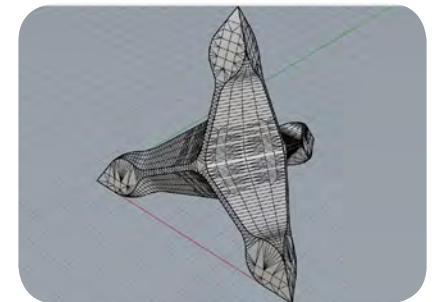
VOM CAD-MODELL ZUM STL

Das gewünschte 3D Objekt kann in einem CAD Programm wie Rhinoceros 3D oder FreeCAD modelliert werden. In den folgenden Abbildungen wird das 3D Objekt mit Rhinoceros 3D bearbeitet. Diese Anleitung startet mit einem fertig modellierten CAD Modell. Wenn man das Modell selbst kreiert so handelt es sich am Ende meist um einen geschlossenen Flächenverband/ closed Polysurface oder eine Leichtbau-Extrusion/lightweight extrusion. Der Unterschied zwischen einem Flächenverband und einer Extrusion macht sich erst ab einer großen Datenmenge bemerkbar. Große Ansammlungen von Flächenverbänden können sehr träge werden und viel Speicherplatz

reaktionsschnell sind und wenig Speicherplatz verbrauchen (McNeel Wiki - Lightweight Extrusion Objects, 2020). Für die Weiterverarbeitung im Slicer benötigen wir allerdings das Dateiformat STL beziehungsweise ein Polygonnetz. Den Flächenverband oder die Extrusion kann man im CAD Programm selbst oder über das Exportieren in das STL Format in ein Polygonnetz umwandeln. Bei der Umwandlung in der CAD Software selbst kann man oft die Anzahl der Polygone, die für die Detailgenauigkeit zuständig sind, selbst definieren. 3D Modelle, welche man von Websites herunterladen kann sind meist schon ein fertiges Polygonnetz.



Mesh with fewer polygons /
 Polygonnetz mit weniger Vielecken

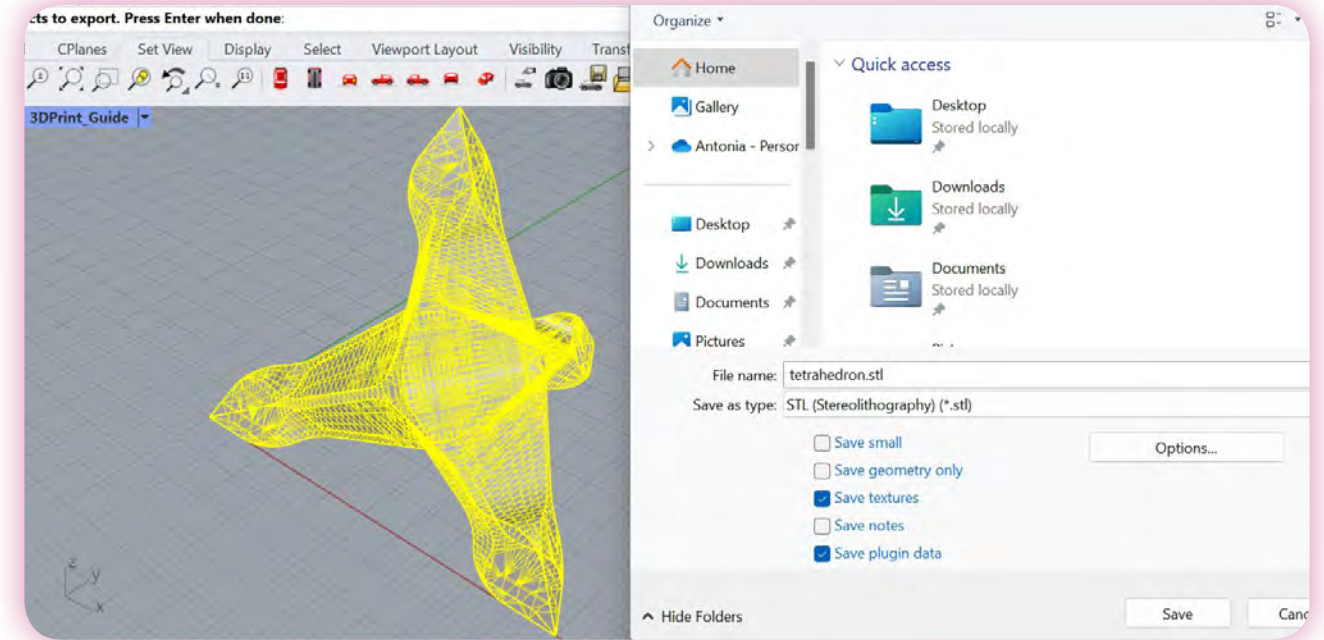


Mesh with more polygons /
 Polygonnetz mit mehreren Vielecken

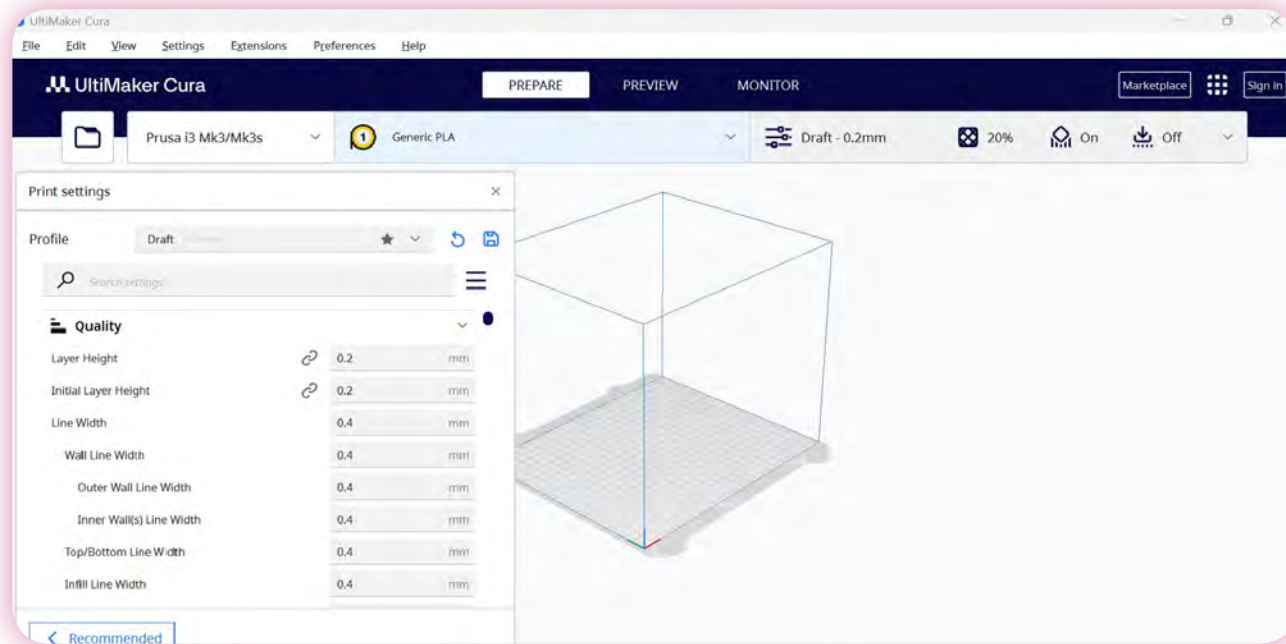
STL TO G-CODE

After the object has been converted into a polygon mesh, you can export and save it as an STL format. Now open the slicer UltiMaker Cura and import the STL. In a slicer like UltiMaker Cura, the object is converted into a G-code, which serves as a kind of roadmap for the printer. Therefore you first have to adjust the settings in the upper bar. On the top left you can select the corresponding FFF printer, in the middle you set the type and color of the material and the diameter of the print nozzle and on the right you select the desired print profile or adjust it according to your requirements. When you

have checked all settings, you can press the slice button at the bottom right. This step does not involve the configuration of any specific print settings. For now, the goal is to get an overview of the entire printing process. Individual print parameters will be discussed separately from page 064.



click on object -> file -> export selected -> save as type: .stl
klicke auf Objekt -> Datei -> exportiere Auswahl -> speichern als Typ: .stl

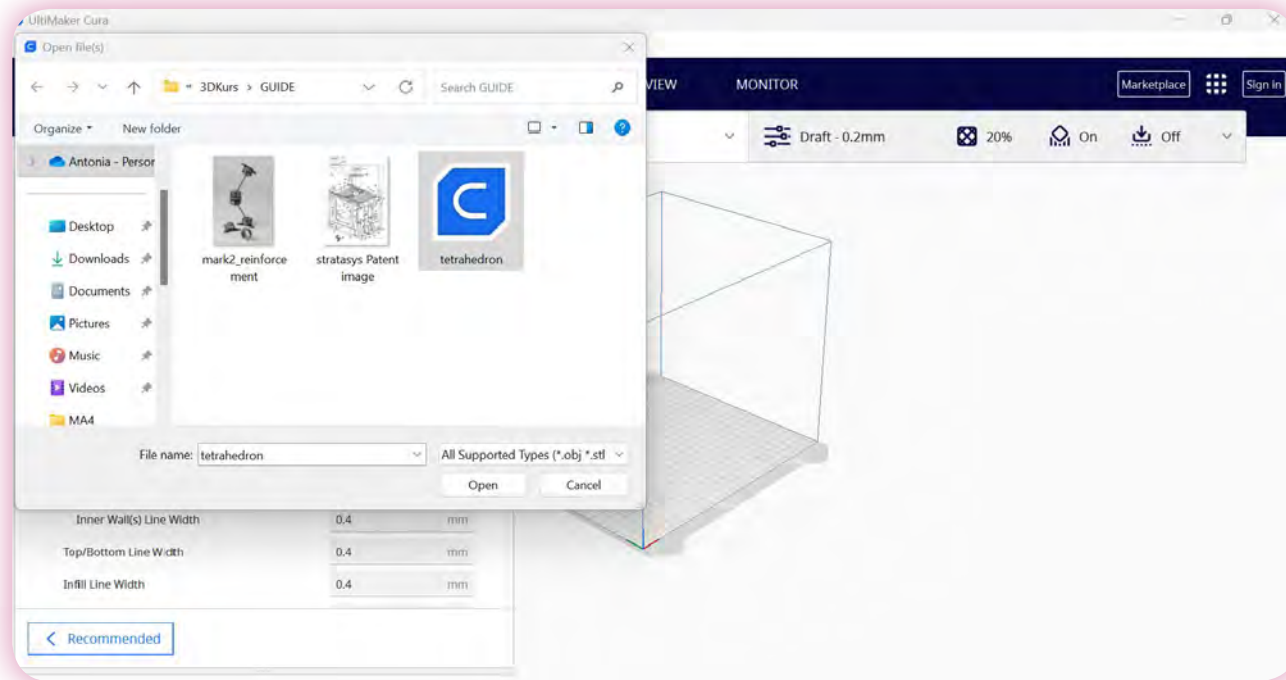


open program Ultimaker Cura
 öffne Programm Ultimaker Cura

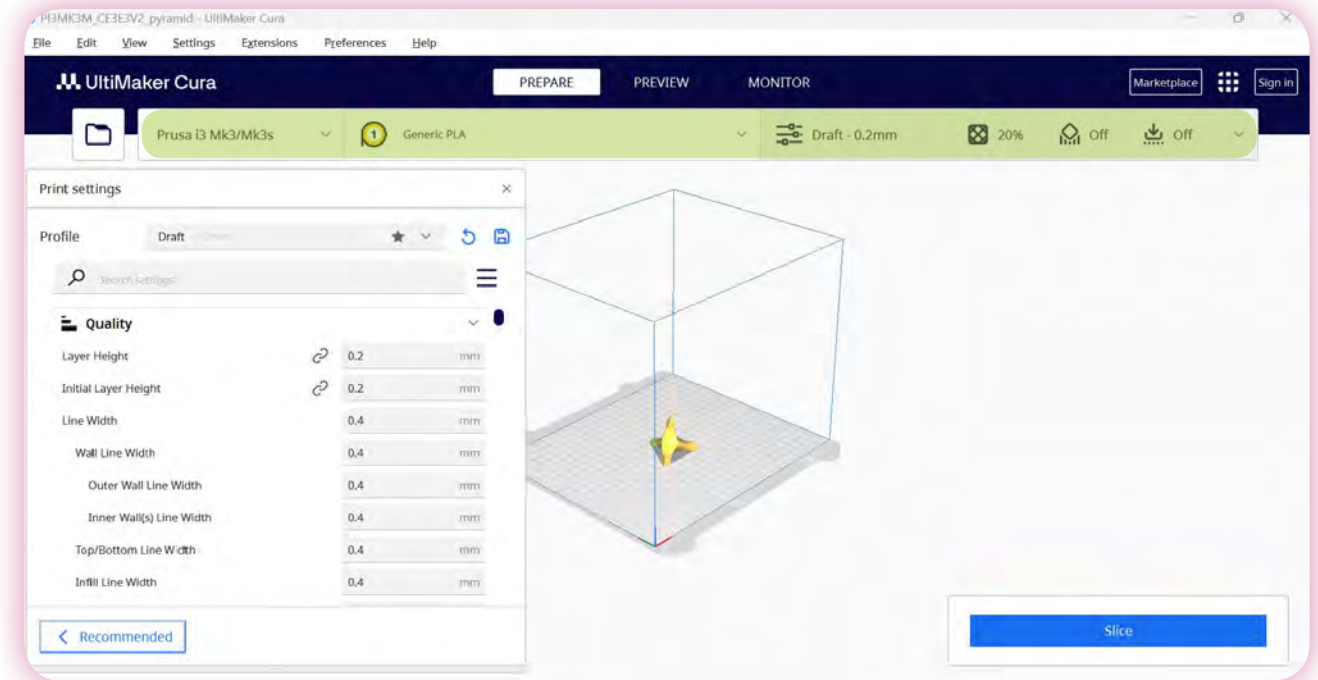
VOM STL ZUM G-CODE

Nachdem das Objekt in ein Polygonnetz umgewandelt wurde, kannst Du es als STL-Format exportieren und speichern. Öffne nun den Slicer Ultimaker Cura und importiere das STL. Das Objekt wird in einem Slicer wie Ultimaker Cura in einen G-Code, welcher als eine Art Fahrplan für den Drucker dient, umgewandelt. Dafür musst Du zunächst die Einstellungen in der oberen Leiste anpassen. Oben links kannst Du den jeweiligen FFF Drucker auswählen, mittig stellst Du Art und Farbe des Materials und den Durchmesser der Druckdüse ein und rechts wählst Du das gewünschte Druckprofil aus beziehungsweise passt es deinen Vorstellungen entsprechend an.

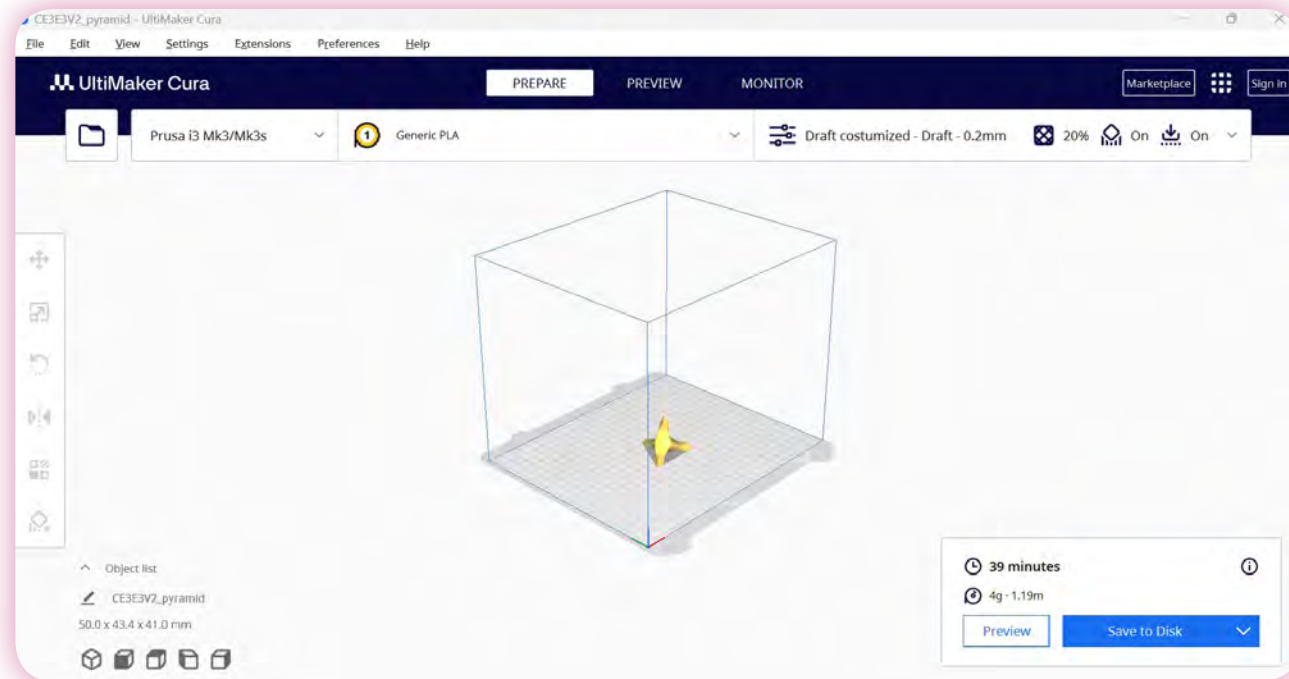
Wenn du alle Einstellungen überprüft hast, kannst du den Slice-Button unten rechts drücken. In diesem Schritt werden zunächst keine spezifischen Druckeinstellungen vorgenommen. Es geht vorerst darum sich einen Überblick über den gesamten Druckprozess zu verschaffen. Auf einzelne Parameter wird dann separat ab Seite 064 eingegangen.



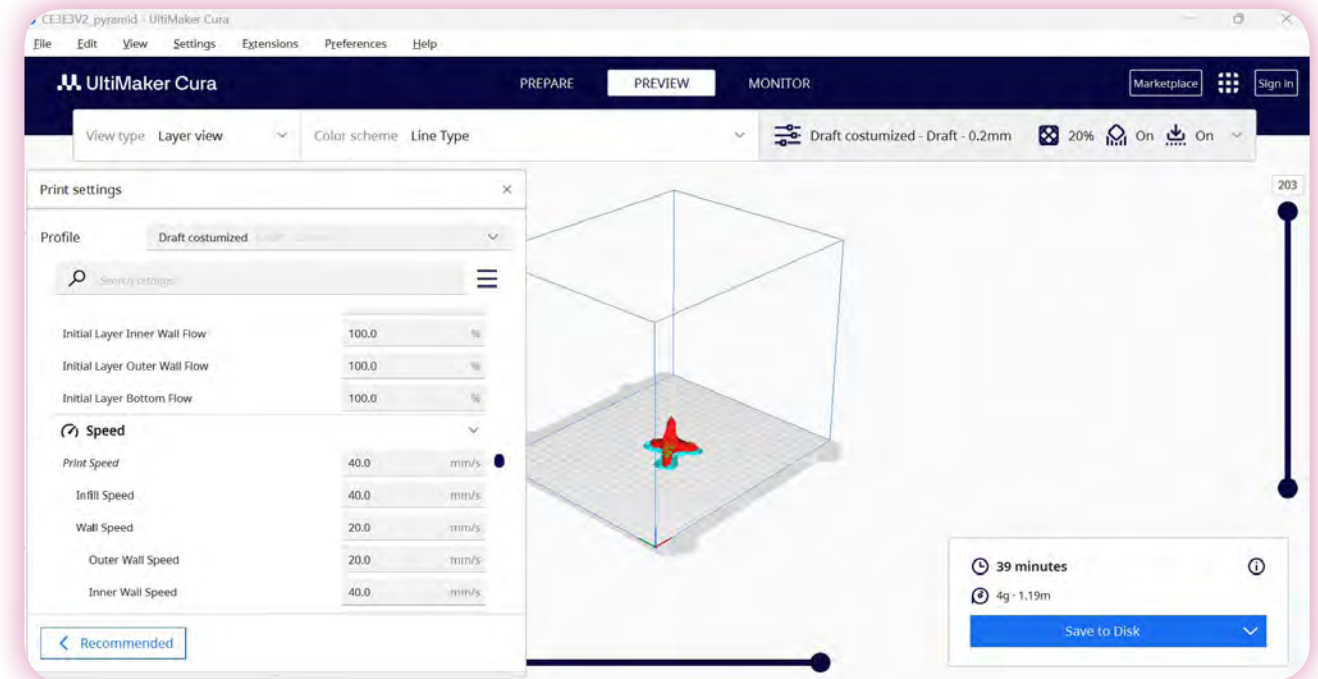
File -> Open File -> choose STL File -> Open
Datei -> Öffne Datei -> wähle STL Datei -> Öffnen



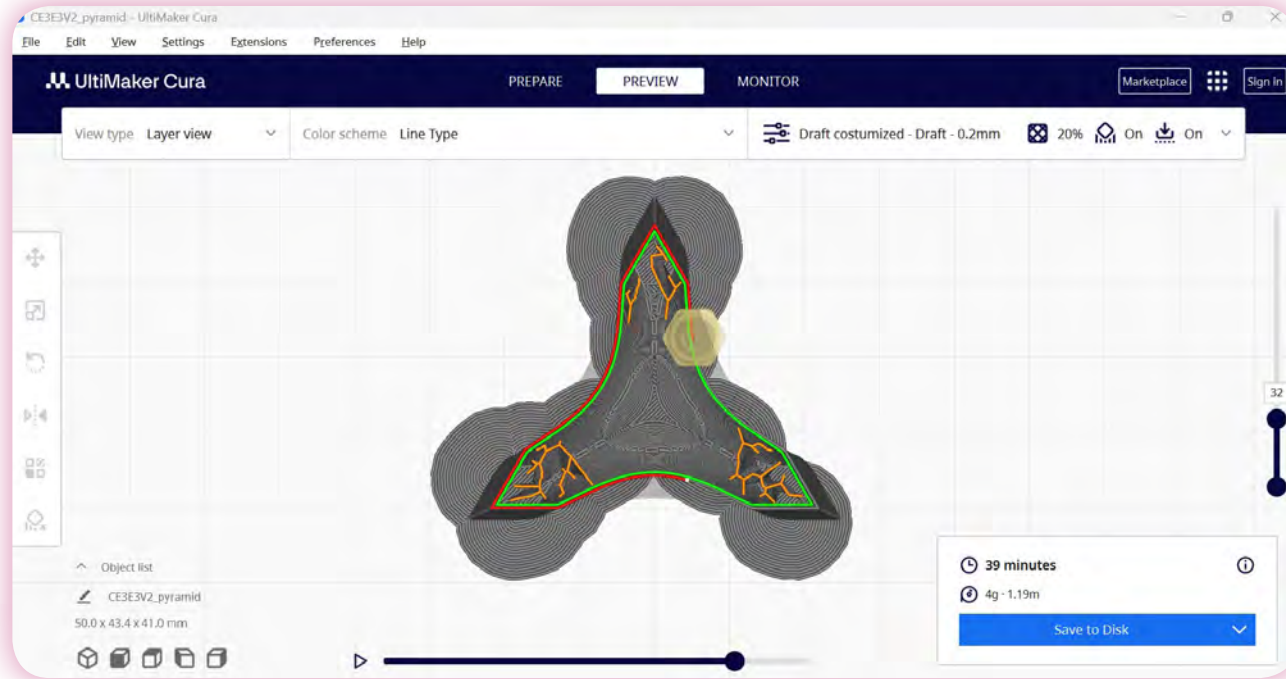
check printer -> check material -> define printing profile -> Slice
klicke auf Objekt -> Datei -> exportiere Auswahl -> speichern als Typ: .stl



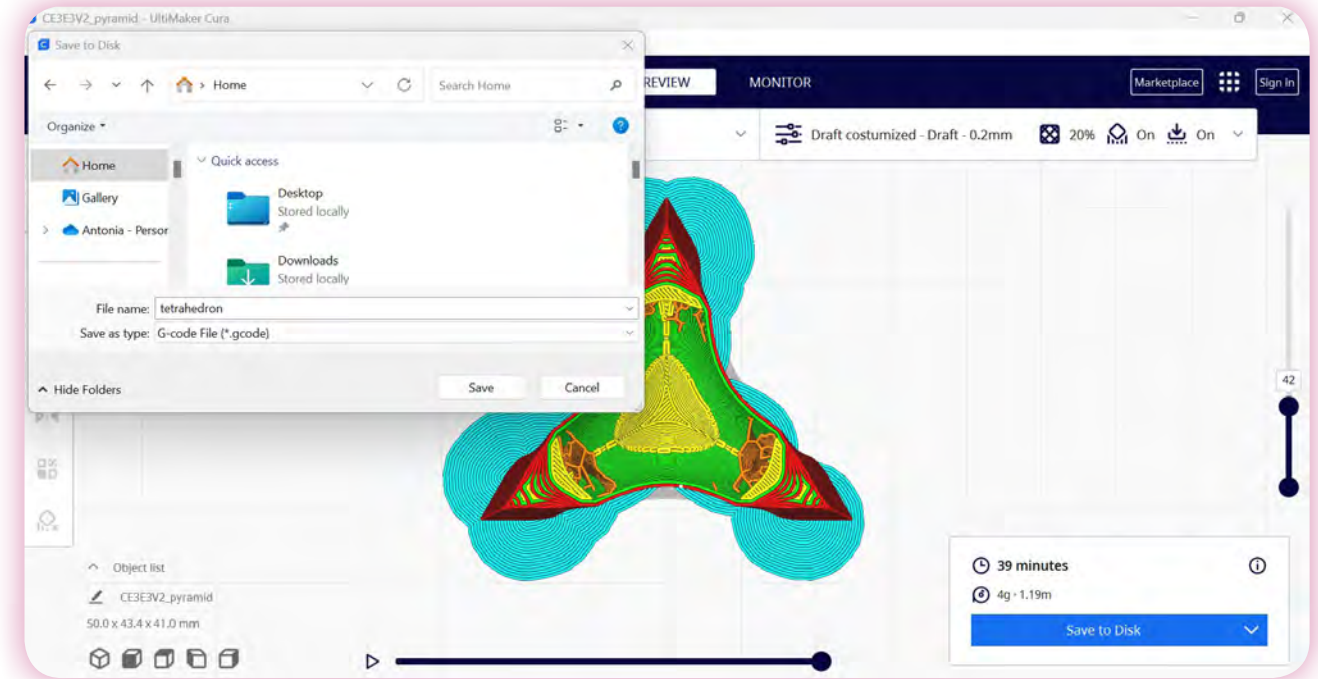
wait until program finished Slicing -> click on Preview Button
 warte bis Programm das Slicen beendet hat -> klicke auf den Preview Button



Slicing View
 Slicing Ansicht



simulate the printing process with sliders -> Bottom Slider for Print Path -> Right Slider for Layers
 Simuliere den Druckprozess mit Reglern -> unterer Regler für Druckpfad -> rechter Regler für Schichten



click on the Save to Disk Button -> save as G-code
 klicke auf den Speichere auf Festplatte Button -> speichere als G-code

PRINTER SETUP



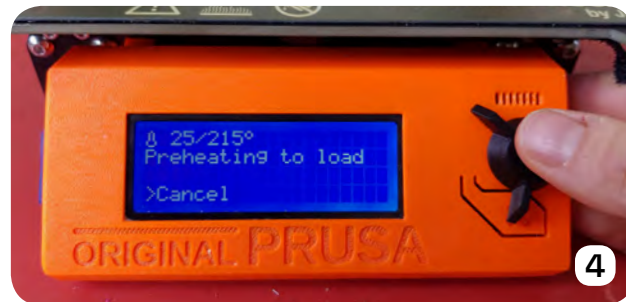
turn on printer
schalte Drucker ein



click on Wheel and choose 'Load Filament'
klicke auf Rad und wähle 'Load Filament'

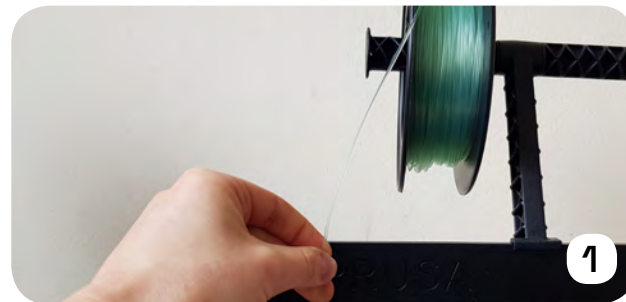


choose material you want to print with
wähle das Material, mit dem du drucken möchtest



Preheating
Vorheizen

DRUCKER VORBEREITEN



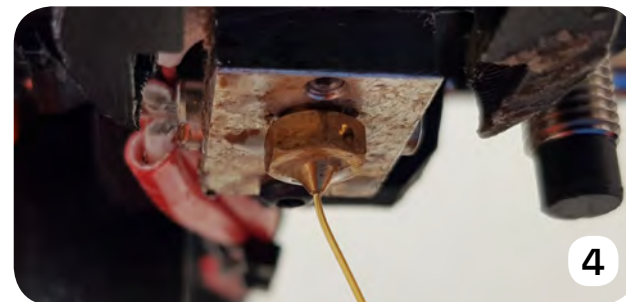
while preheating, insert Filament in feeder
während des Vorheizens Filament in Feeder einführen



clean nozzle
Düse reinigen



wait for beeping sound & click on Wheel
Piepton abwarten & Rad klicken



check if Filament extrudes right
prüfen, ob Filament richtig extrudiert wird

BED LEVELING



The print bed levelling is about calibrating the parallelism and height between the print nozzle and the entire print surface. The calibration affects the adhesion and aesthetics of the print. As a print bed is connected to the rest of the printer hardware either via adjustment screws or rails, the type of leveling differs. If a print bed is calibrated via 3 or 4 adjusting screws, the height of the nozzle and the parallelism of the 3 axes must be adjusted. A piece of paper is usually placed between the nozzle and the print surface and the print head is moved to all 4 corners of the print bed. The piece of paper is then pushed back and forth and you have to try to create the same resistance in all corners by feel.

The big disadvantage of this is that you have to rely on your gut feeling, which has to be built up through experience. With a print bed that runs on rails, you only have to adjust the height of the print nozzle. Depending on the model, this setting can be found on the printer display under designations such as 'z-offset' or 'First layer calibration'. As an example of print bed leveling, the so-called 'First layer calibration' is carried out on the Prusa in the illustrations on these pages.

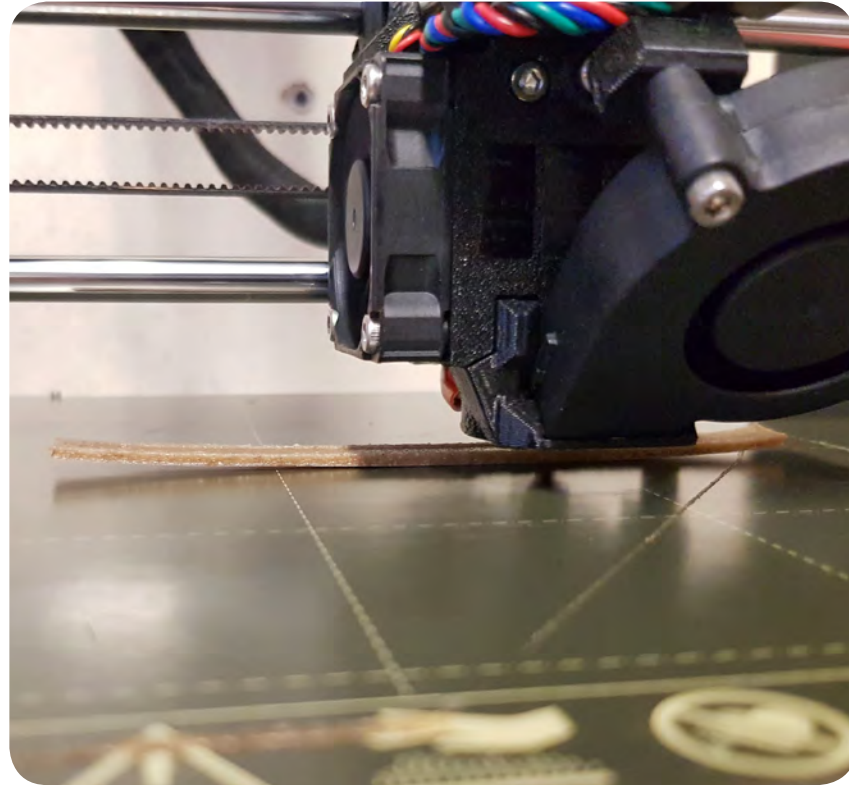
DRUCKBETT-NIVELLIERUNG

Bei der Druckbett-Nivellierung geht es darum die Parallelität und die Höhe zwischen der Druckdüse und der gesamten Druckfläche zu kalibrieren. Die Kalibrierung beeinflusst die Haftung und die Ästhetik des Drucks. Da ein Druckbett entweder über Einstellschrauben oder Schienen mit der restlichen Druckerhardware verbunden ist, unterscheidet sich hier auch die Art der Nivellierung. Wenn ein Druckbett über 3 oder 4 Einstellschrauben kalibriert wird, so muss man sowohl die Höhe der Düse als auch die Parallelität der 3 Achsen einstellen. Dabei wird meist ein Stück Papier zwischen Düse und Druckbett geschoben und der Druckkopf in alle 4 Ecken des Druckbetts bewegt.

Man schiebt dann das Stück Papier hin und her und muss versuchen nach Gefühl den gleichen Widerstand in allen Ecken zu erzeugen. Der große Nachteil dabei ist, dass man sich auf sein Gefühl verlassen muss, welches erst durch Erfahrungen aufgebaut werden muss. Bei einem Druckbett, welches auf Schienen läuft muss man nur die Höhe der Druckdüse anpassen. Diese Einstellung findet man am Druckerdisplay nach Modell unter Bezeichnungen wie 'z-offset' oder 'First layer calibration'. Stellvertretend für die Druckbett-Nivellierung wird in den Abbildungen auf diesen Seiten die sogenannte 'First layer calibration' am Prusa durchgeführt.



ADHEASION



It depends on the print geometry, the print settings and the printer model whether the print bed needs to be prepared to ensure good adhesion of the first layers. If the print geometry becomes tapered in the upper layers, the risk of the object detaching during printing is low. If the volume of the object expands in the upper layers, there is a higher risk that it will detach from the print bed during the printing process. The detachment is triggered by the print nozzle, which slightly moves the already printed mass when applying new layers.

HAFTUNG

In wiefern das Druckbett vorbereitet werden muss, um eine gute Haftung der ersten Schichten zu gewährleisten, hängt stark von der Druckgeometrie, den Druckeinstellungen und des Druckmodells ab. Verjüngt sich die Druckgeometrie in den höheren Schichten, so ist die Wahrscheinlichkeit gering, dass sich das Objekt während des Drucks ablöst. Erweitert sich das Volumen des Objektes in den höheren Schichten, so ist die Wahrscheinlichkeit höher, dass es sich noch während des Druckvorgangs vom Druckbett ablöst. Das Ablösen wird durch die Druckdüse ausgelöst, welche beim Auftragen der Schichten die bereits gedruckte Masse immer ein wenig in Bewegung versetzt.

Die Druckeinstellungen im Slicer können die Haftung verschlechtern, wenn die Druckdüse beispielsweise bei einer länglichen Zugprobe immer nur von Links nach rechts fährt und sich dabei die äußeren Kanten mit der Zeit vom Druckbett lösen. Wenn der Druckkopf hingegen mehrere Richtungswechsel während des Drucks einschlägt beeinflusst das in der Regel die Haftung weniger. Andere Druckparameter, welche die Haftung beeinflussen sind die Druckbetttemperatur und die Kühlungseinstellungen. Die Druckbetttemperatur liegt in der Regel bei etwa 60°C und kann für die erste Schicht auch gern ein wenig erhöht werden.

Ein voll aufgedrehter Kühllüfter kann dabei die Betttemperatur negativ beeinflussen und sollte sich daher erst nach 3-4 Schichten anstellen. Viele Drucker verfügen über eine Druckplatte aus Glas, oft auch mit haftverstärkender Beschichtung. Andere Drucker sind mit einer magnetischen Federstahl-druckplatte ausgestattet, welche vom Drucker gelöst werden kann.

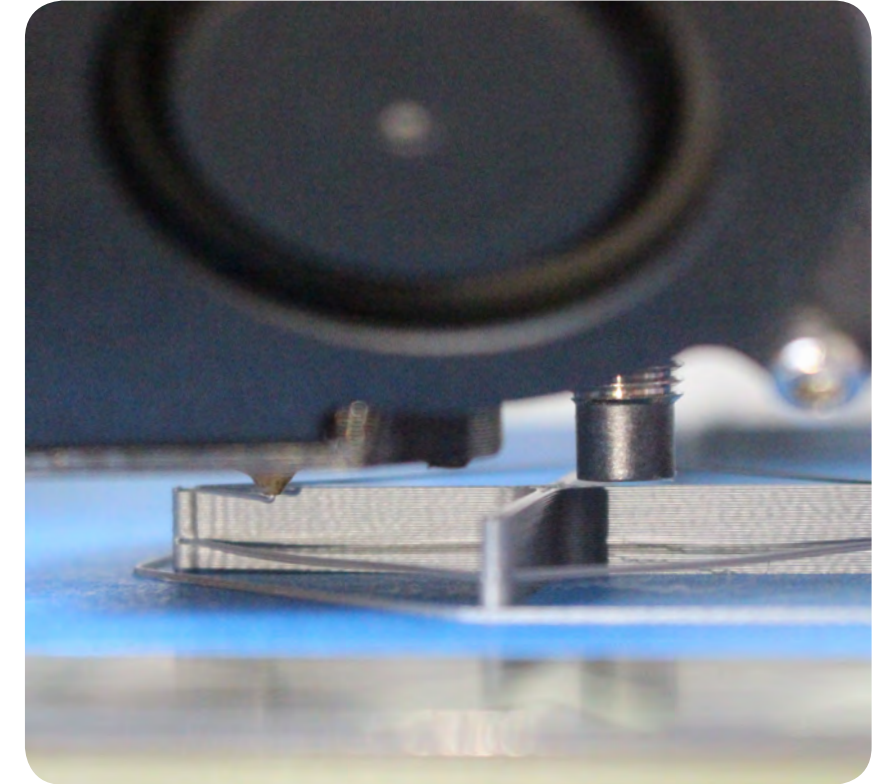
ADHEASION

The print settings in the slicer can also affect adhesion. If the nozzle only moves from left to right, to print for example an elongated tensile test sample, the outer edges will detach from the print bed over time (see figure). If, on the other hand, the print head often changes direction during printing, this generally has less influence on adhesion. Other print parameters that influence adhesion are the print bed temperature and the cooling settings. The print bed temperature is usually around 60°C and can also be increased a bit for the first layer. A fully turned on cooling fan can have a negative effect on the bed temperature in the first layers and should therefore only be turned on after 3-4 layers.

Many printers have a printing bed made of glass, often also with an adhesion-enhancing coating. Other printers are equipped with a magnetic spring steel plate that can be detached from the printer. To release the print object carefully from the plate without damaging it, the plate can be bent slightly. Experience has shown that these print beds are easier to handle. You can counteract the detachment of the print object during the printing process by roughening the print surface or by applying a sticky substance to the print surface. Four options for preparing the print bed are presented on the next page.

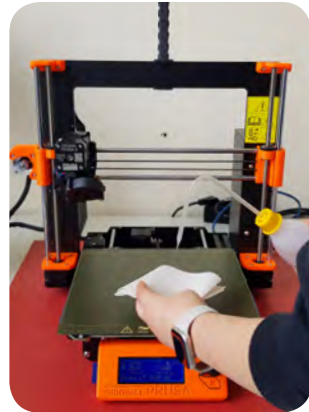
HAFTUNG

Um das Druckobjekt behutsam von der Platte zu lösen ohne es zu beschädigen, kann die Platte leicht gebogen werden. Erfahrungsgemäß sind diese Druckbetten leichter zu handhaben. Man kann dem Ablösen des Druckobjektes während des Druckvorgangs mit einer Verrauung des Untergrundes entgegenwirken oder indem man eine klebrige Substanz auf die Druckfläche aufbringt. Auf der nächsten Seite werden vier Möglichkeiten zur Vorbereitung des Druckbetts vorgestellt.



PREPARE PRINTBED

ethanol



- + fast application
- + affordable
- low chance of success
- + smooth printing surface
- + print bed does not need to be cleaned after use

4/5 *

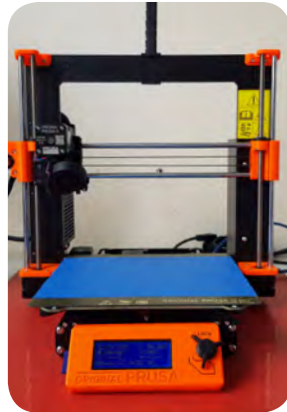
gluestick



- + fast application
- + affordable
- medium chance of success
- sticky print surface
- print bed must be cleaned after use

2/5 *

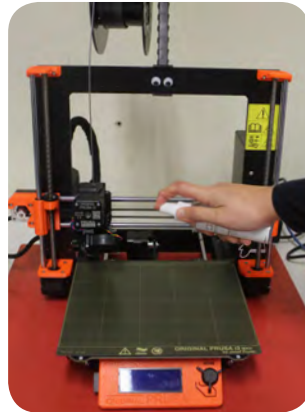
blue tape



- time-consuming preparation
- + affordable
- + high chance of success
- unevenness in print bed
- + print bed does not need to be cleaned after use

3/5 *

adhesive spray

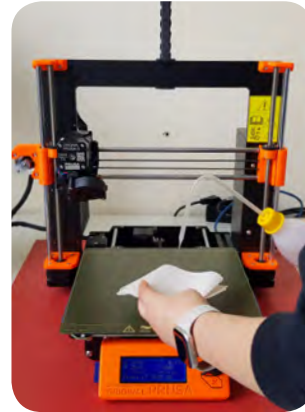


- + fast application
- expensive
- + high chance of success
- + smooth printing surface
- print bed must be cleaned after use

3/5 *

DRUCKBETT VORBEREITEN

Ethanol



- + schnelle Anwendung
- + erschwinglich
- niedrige Erfolgschance
- + glatte Druckfläche
- + Druckbett muss nicht nach Anwendung gereinigt werden

4/5 *

Klebestift



- + schnelle Anwendung
- + erschwinglich
- mittlere Erfolgschance
- klebrige Druckfläche
- Druckbett muss nach Anwendung gereinigt werden

2/5 *

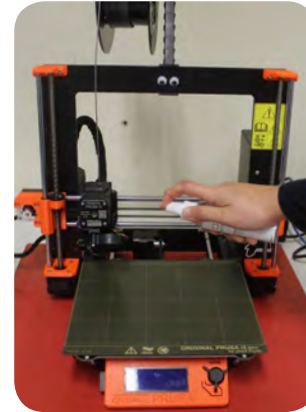
Blue Tape



- aufwendige Vorbereitung
- + erschwinglich
- + hohe Erfolgschance
- Unebenheiten im Druckbett
- + Druckbetts muss nicht nach Anwendung gereinigt werden

3/5 *

Haftspray



- + schnelle Anwendung
- kostenspielig
- + hohe Erfolgschance
- + glatte Druckfläche
- Druckbett muss nach Anwendung gereinigt werden

3/5 *

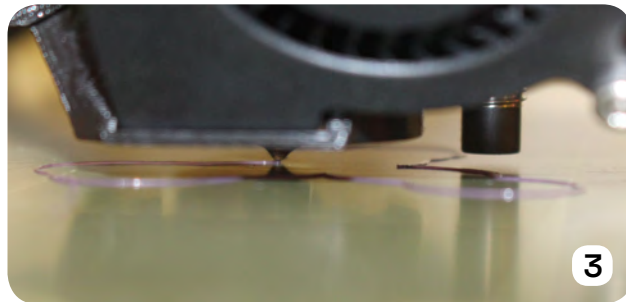
START PRINT



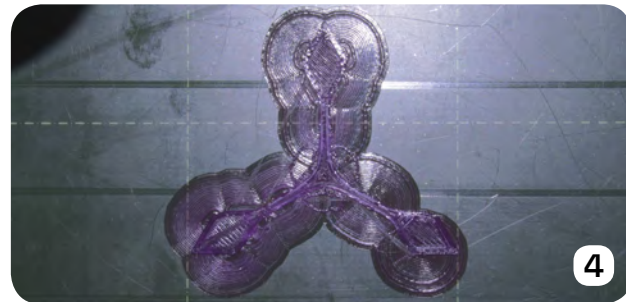
upload G-Code on SD card -> insert SD card
G-Code auf SD-Karte laden -> SD-Karte einstecken



choose file -> wait for heating
Datei wählen -> Aufheizen abwarten

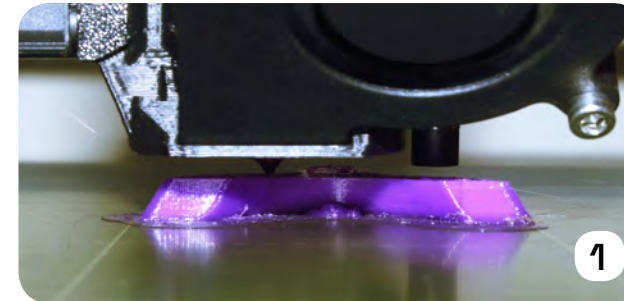


3D Printer starts printing
3D Drucker startet den Druck

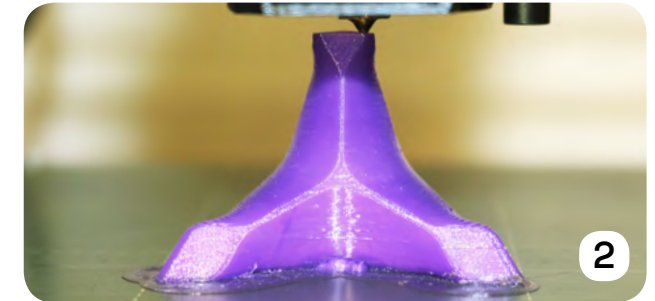


check adhesion of 1st layer
Haftung der 1. Schicht prüfen

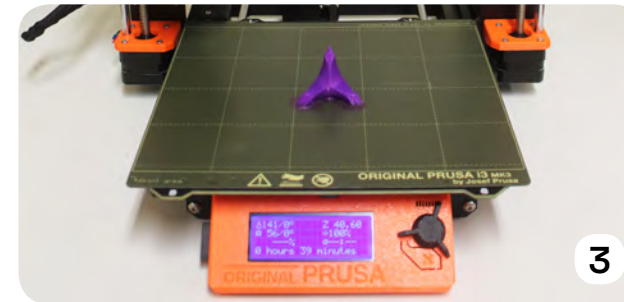
STARTE DRUCK



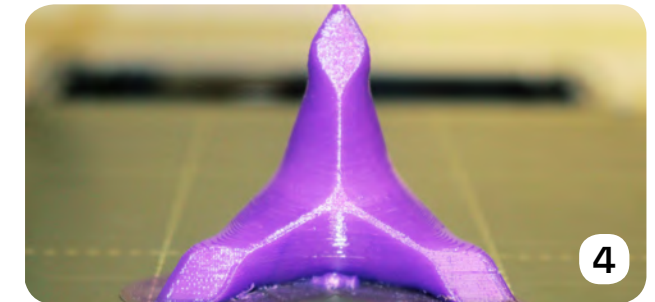
check adhesion during first layers
beobachte Haftung der ersten Schichten



wait...
warte...

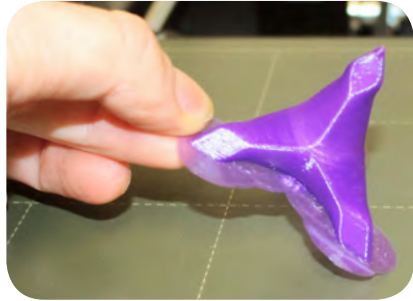


and Tadaaa!
und Tadaaa!



Congratulations to your first 3D print :)
Glückwunsch zu Deinem ersten 3D-Druck :)

POST PROCESSING



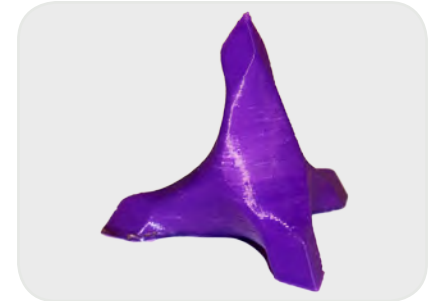
When all the printer components have cooled down, you can remove the object from the print bed and admire your first 3D print. Don't worry if the object does not yet meet your aesthetic requirements. Just play around with the print parameters and you'll soon fulfill your expectations. If there is still support material on your object, you can carefully remove it. The surface can be smoothed with sandpaper or acetone for further post-processing. You can join objects together using a 3D printing pen, a soldering iron or glue.

If you wish, you can also give the object a new coat of paint or apply patterns using water transfer printing. During post-processing, make sure that PLA, for example, can no longer be recycled as soon as you connect it to another material such as epoxy resin or paint. Work carefully to avoid large quantities of misprints and try to make your projects sustainable.

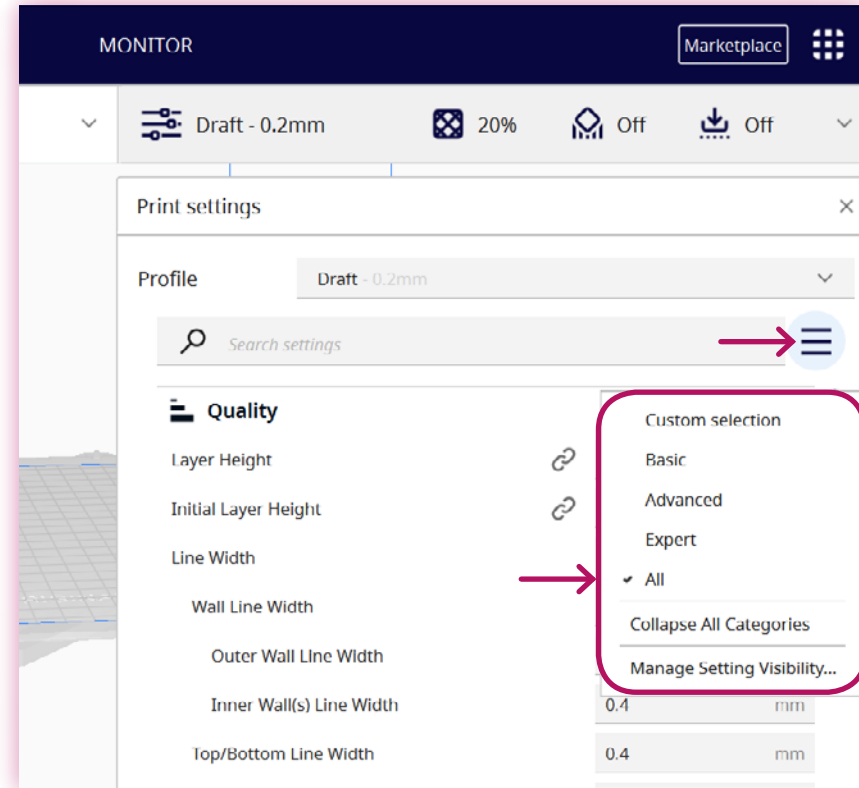
NACHBEARBEITUNG

Wenn alle Druckerbauteile abgekühlt sind, kannst Du das Objekt vom Druckbett lösen und deinen ersten 3D Druck bestaunen. Ärgere Dich nicht, wenn das Objekt noch nicht Deinen ästhetischen Anforderungen entspricht. Spiele einfach noch ein wenig mit den Druckparametern herum und Du wirst bald Deinen Anforderungen gerecht werden. Wenn sich noch Stützmaterial an Deinem Objekt befindet, kannst du es vorsichtig entfernen. Zur weiteren Nachbearbeitung lässt sich die Oberfläche mit Schleifpapier oder Aceton glätten. Objekte kannst Du mithilfe von einem 3D Druckstift, einem Lötkolbens oder Kleber miteinander verbinden.

Nach Belieben kann man dem Objekt natürlich auch farblich einen neuen Anstrich verleihen oder mit Wassertransferdruck Muster auftragen. Achte bei der Nachbearbeitung darauf, dass sich zum Beispiel PLA nicht mehr recyceln lässt, sobald Du es mit einem anderen Material wie Epoxidharz oder Lack in Verbindung bringst. Arbeite sorgfältig, um große Mengen an Fehldrucken zu vermeiden und versuche deine Projekte nachhaltig zu gestalten.



SLICER - RELEVANT PARAMETERS



To finish up this introduction to 3D printing, let's take a look at the relevant print parameters that experience has shown to be useful for most projects. The following best-of print parameters should help you to get a rough overview instead of being intimidated by the numerous settings. To make all settings visible in the slicer, you can click on the three horizontal lines in the print settings window and select 'All'. You don't have to research which parameters are responsible for what and how they influence the printing process. Therefore there is the additional information plug-in 'Settings Guide' in the Ultimaker Market Place. The following pages show excerpts from this plug-in.

SLICER - RELEVANTE PARAMETER

Zu guter Letzt möchte ich nochmal auf relevante Druckparameter eingehen, welche sich erfahrungsgemäß für die meisten Projekte als nützlich erweisen. Das folgende Best-of an Druckparametern soll dabei helfen sich nicht von den zahlreichen Einstellungen einschüchtern zu lassen, sondern einen groben Überblick zu erhalten. Um alle Einstellungen im Slicer sichtbar zu machen, kannst Du im Fenster für Druckeinstellungen auf die drei waagerechten Striche klicken und 'All' auswählen.

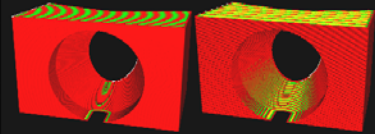
Du musst nicht selbst recherchieren wofür jeder einzelne Parameter zuständig ist und wie er den Druckprozess beeinflusst, dafür gibt es das zusätzliche Informations Plug-in 'Settings Guide' im Ultimaker Market Place. Die folgenden Seiten zeigen Ausschnitte aus diesem Plug-in.

Da der 'Settings Guide' größtenteils nur in englischer Sprache zur Verfügung gestellt wird, endet hier nun die deutsche Übersetzung.

QUALITY- LAYER HEIGHT

Layer Height

The 3D printer puts down plastic in layers. The layer height is the thickness of these layers in millimetres. It is the most important factor in both the visual quality of your final print and the printing time.



Layer height is the single most important setting to affect the overall quality and conversely the printing time. These are just some of the effects:

- Having thinner layers will increase the visual quality of the print. Since the layers are thinner, the stair stepping effect at the layer's borders will be reduced. Also, the layers will be closer together and so the creases in between the layers will be smaller, leading to an overall smoother finish.
- Having thinner layers allows the printer to produce more detail on the top and bottom sides of your print.
- Having thicker layers tends to make the print stronger, to a point. There will be fewer borders between the layers, which tend to be a weak point. The thicker layers will not shear as much.
- Having thicker layers will reduce printing time, because the nozzle will not have to make as many horizontal movements.

Layer height vs. profiles

Many settings depend on the layer height. Because layer height significantly affects the flow rate of material through the nozzle, many parameters of the printing process will change. This is very complex. For instance, when increasing the layer height, you should probably increase the printing temperature slightly to offset the extra rate of heat loss. The temperature then affects the liquidity of the material which will affect how sharp your corners will be and what cooling is needed, and so on. It is always wise to start from a pre-made quality profile available to your printer that has a layer height close to what you want.

You can choose a desired layer height in Custom Mode, but pre-made profiles are available with various layer heights as well. You can choose from profiles for various layer heights in the Recommended Mode using a slider or using the drop-down widget in Custom Mode. Since these profiles also change some parameters that depend on the layer height, you'll probably achieve a better quality that way.

Additional notes

At very low layer heights you could run into the resolution limit of your Z axis. Look up the step size of your printer's Z axis and make sure that the layer height is a multiple of this. If it doesn't match up properly, some layers will be thicker than others which will result in banding.

Note that the layer height setting does not hold for the initial layer of the print or for the raft layers, which have their own settings to adjust the layer height separately. When using adaptive layers, this layer height setting will be used as a baseline but the actual layer height will have some variation.

Normal - 0.15mm 0% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality

Layer Height	0.15	mm
Initial Layer Height	0.2	mm
Line Width	0.4	mm
Wall Line Width	0.4	mm
Top/Bottom Line Width	0.4	mm

Walls

Top/Bottom

Infill

Material

Speed

Travel

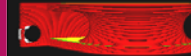
Cooling

Support

QUALITY - INITIAL LAYER HEIGHT

Initial Layer Height

This setting defines the thickness of the first layer of your print. The initial layer is normally printed thicker than the rest in order to create a stronger adhesion with the build plate. With this setting, the initial layer's thickness can be increased without reducing the resolution of the rest of the print.



Increasing the initial layer's thickness causes the nozzle to extrude more material over the same distance. This takes extra force, as the material spreads out to the sides to fill the full line width. This extra force causes the material to stick better to the build plate. Additionally, the thicker layer will catch any irregularities in the flatness of the surface. If the build plate is slightly bent, the variability will be absorbed by the thickness of the first layer, whereas otherwise the nozzle could scrape it off in the second layer.

Having too thick of an initial layer causes the first layer to sag more, which causes elephant's feet. The Initial Layer Horizontal Expansion setting can prevent this by giving it a small negative value.

Affects

- Regular Fan Speed at Height
- Regular Fan Speed at Layer
- Raft Base Thickness

Normal - 0.15mm 0% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality

Layer Height	0.15	mm
Initial Layer Height	0.2	mm
Line Width	0.4	mm
Wall Line Width	0.4	mm
Top/Bottom Line Width	0.4	mm

Walls

Top/Bottom

Infill

Material

Speed

Travel

Cooling

Support

QUALITY - LINE WIDTH

Line Width

This is the horizontal width of the lines that the printer will place down. Normally the diameter of the nozzle opening determines how wide your lines will be, but by extruding more or less material, the printer can vary a little bit in how wide the lines will become.



Reducing the line width allows the printer to print more details. In particular it also allows the printer to print thin parts. Line width is one of the most influential settings in your print. Here are some of the effects:

- Printing thinner lines will allow thinner pieces to be printed, since it can even fit a line in the thinnest parts.
- Adjusting the line width to an even multiple of the thickness of your print can make the object stronger and make the material flow better.
- A smaller line width will make your top surface look smoother.
- Printing lines slightly smaller than your nozzle size tends to improve strength. It allows the nozzle to fuse adjacent lines together when it makes a second pass slightly over the previous line.
- Printing lines that are too wide will lead to underextrusion. The printer will attempt to extrude more material, enough to fill the desired width of the line. That material will attempt to flow in whichever direction it can. However at some point the back pressure will become too great such that the material will no longer flow all the way to the side of the very wide lines. This then leaves gaps between the lines.
- Printing lines that are too small will also lead to underextrusion. If the material doesn't flow fast enough through the nozzle, the surface tension of the material will cause it to coagulate into small droplets, making the extrusion uneven and leaving gaps in between the droplets.
- Printing thinner lines will considerably increase the printing time.

It's not advisable to reduce the line width below 60% of the nozzle size or above 150%. Both may fail to extrude enough material.

In thin parts, the line width will automatically be adjusted to make it fit the local width of the part there. There is no need to ensure that the part's width is a multiple of the line width. The Minimum Wall Line Width determines how far they can get adjusted in each direction.

Keeping the flow constant

Great fluctuations in flow are sometimes problematic for FDM printers. The nozzle chamber keeps some material under pressure, which causes the actual flow rate out the nozzle to be delayed. It'll take a while for the flow rate to increase or decrease. Printers with a Bowden system to feed the filament also have springiness in the Bowden tube, which makes the effect much worse. As a result of this, you'll get underextrusion when switching to a higher flow rate and overextrusion when switching to a lower flow rate. Therefore it is a good idea to keep the flow rate as constant as possible.

Normal - 0.15mm 0% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality

Layer Height	0.15	mm
Initial Layer Height	0.2	mm
Line Width	0.4	mm
Wall Line Width	0.4	mm
Top/Bottom Line Width	0.4	mm

Walls

Top/Bottom

Infill

Material

Speed

Travel

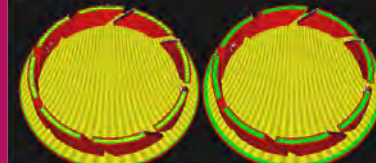
Cooling

Support

WALLS - THICKNESS

Wall Thickness

This setting determines how thick the wall around the print will be. Ultimately it adjusts the number of inner walls to achieve the desired thickness.



The wall thickness must be a multiple of the wall line width. If it is not, it will be rounded to be an exact multiple. Keep in mind though that one of the walls will be an outer wall, and the rest will be inner walls. These walls can have different line widths.

The thickness of the walls is a major factor in how strong the print will be. Since the walls are adjacent, they can reinforce each other leading to a stronger part. For larger prints, this can be a much more effective way to get a strong object than adjusting the infill, depending on the shape.

Increasing the wall thickness will:

- Greatly increase the strength of the print.
- Reduce the shine-through effect where the infill pattern is visible on the outside.
- Improve overhangs, because the wall lines are usually more oriented towards their nearest resting point.
- Make the model watertight more easily.
- Greatly increase the printing time and material usage of the print.

Affects

- Wall Line Count

Affect ed by

- Outer Wall Line Width
- Spiralize Outer Contour

Normal - 0.15mm 0% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality

Walls

Wall Thickness 0.8 mm

Wall Line Count 2

Optimize Wall Printing Order

Top/Bottom

Infill

Material

Speed

Travel

Cooling

Support

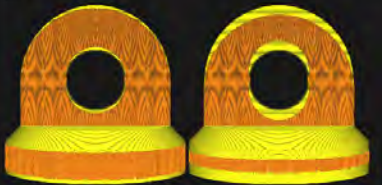
Build Plate Adhesion

Experimental

TOP/BOTTOM - THICKNESS

Top/Bottom Thickness

With the Top/Bottom Thickness you can adjust how thick of a solid skin is created on the top and bottom sides of your print.



This affects both the top and bottom sides of the print. The top and bottom can also be configured individually with the Bottom Thickness settings.

If this setting is not a multiple of the layer height, it will get rounded to the nearest multiple. This setting only determines how many top and bottom layers will be generated.

Greater top and bottom thickness will have several effects, beneficial for total quality but detrimental to productivity.

- Your part will be stronger. Greater top and bottom thickness will mean that a smaller part of your print will get filled with the sparse infill pattern. Instead it gets made completely solid.
- The top surface will be smoother. Any pilling and sagging in the top skin will get smoothed out in the layers put on top.
- It's more likely that your model will be water-tight. Both on the top and bottom sides there will be more layers to block water or other fluids from getting in, and to recover from sagging overhangs.
- Your print will use more material, since more layers are filled completely.
- Your print will take longer, since more material must be placed down and these layers are normally printed more slowly than the infill.

Affects

- Top Thickness
- Bottom Thickness

Normal - 0.15mm 0% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality Walls Top/Bottom

Top/Bottom Thickness 0.8 mm

Top Thickness 0.8 mm

Top Layers 6

Bottom Thickness 0.8 mm

Bottom Layers 6

Top/Bottom Pattern Lines

Monotonic Top/Bottom Order ☐

Enable Ironing ☐

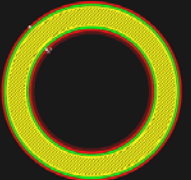
Infill Material Speed

TOP/BOTTOM - PATTERN

Top/Bottom Pattern

This setting allows you to choose how the top and bottom surfaces get filled up with material. Several different printing patterns are available, but the patterns here are more limited than for infill. Only patterns that create solid layers are available.

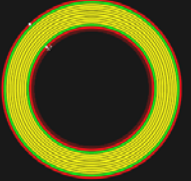
Lines



The basic lines pattern draws straight lines across the surface. These lines are oriented by default such that they are well supported by the infill and support. The direction of the lines alternates between layers.

- Provides a nice surface quality.
- Very rigid in the directions that the lines are drawn.
- Adheres strongly to the walls, creating relatively strong parts.

Concentric



The concentric pattern continues drawing contours towards the inside of the model.

- Equally strong in all directions.
- Prevents creating air pockets and gaps. It's easier to create water-tight objects with this pattern.
- Great overhang quality, because the lines tend to bridge very well.
- If the part is circular, this can create a nasty spot in the centre where the contours converge.

Normal - 0.15mm 0% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality Walls Top/Bottom

Top/Bottom Thickness 0.8 mm

Top Thickness 0.8 mm

Top Layers 6

Bottom Thickness 0.8 mm

Bottom Layers 6

Top/Bottom Pattern Lines

Monotonic Top/Bottom Order ☐

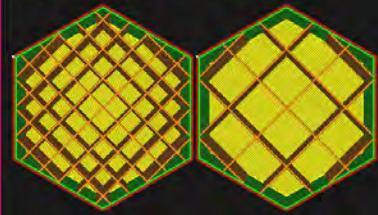
Enable Ironing ☐

Infill Material Speed

INFILL - DENSITY

Infill Density

This setting configures the density of the volume inside the print, which is a major factor in the strength of the final print as well as the top surface quality. The greater the infill density, the closer the infill lines will be placed together. You can even go above 100% density, but that will result in overextrusion.



Different densities work better with different infill patterns. Infill patterns with lots of corners and lots of crossings will not work well at great infill densities. Corners are a problem because the filament tends to drag along with the corner, creating air pockets in the outside of the corner where the material should've been deposited. Crossings are an even greater problem, because when one line crosses another it's flow will get interrupted, causing underextrusion right after the crossing.

Increasing the infill density (by reducing the line distance) has a big effect on your print, namely:

- Your print will be stronger.
- The top surface will be supported better, making it smoother and more water tight.
- Reduced pillowing because the pockets of heat will be smaller.
- Your print will require more material and as a result will be heavier.
- It takes longer to print.

Affects

- Top Layers
- Bottom Layers
- Infill Line Distance
- Infill Pattern
- Infill Overlap Percentage
- Infill Overlap

Normal - 0.15mm 10% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality Walls Top/Bottom Infill

Infill Density 10.0 %

Infill Line Distance 8.0 mm

Infill Pattern Grid

Connect Infill Lines

Infill Line Directions []

Infill Line Multiplier 1

Infill Overlap Percentage 10.0 %

Infill Layer Thickness 0.15 mm

Gradual Infill Steps 0

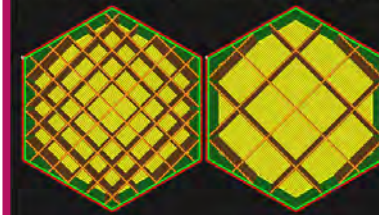
Material

INFILL - LINE DISTANCE

Infill Line Distance

Instead of setting the Infill Density as a percentage, it is also possible to configure infill density by setting the distance between adjacent infill lines. A greater distance between infill lines will result in a lower infill density overall.

Infill Line Distance is a different way of looking at this property than density. The distance between infill lines gives an indication of the distance that the top surface lines have to cross from one infill line to the next. Reducing the Infill Line Distance thus reduces the bridged distance and improves the quality of top surfaces.



Normally the Infill Line Distance is computed from the desired infill density, depending on the selected infill pattern and line width. The Infill Line Distance is leading.

Increasing the infill density (by reducing the line distance) has a big effect on your print, namely:

- Your print will be stronger.
- The top surface will be supported better, making it smoother and more water tight.
- Reduced pillowing because the pockets of heat will be smaller.
- Your print will require more material and as a result will be heavier.
- It takes longer to print.

Affects

- Cross 3D Pocket Size

Affect ed By

- Infill Pattern
- Infill Density
- Infill Line Width

Normal - 0.15mm 10% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality Walls Top/Bottom Infill

Infill Density 10.0 %

Infill Line Distance 8.0 mm

Infill Pattern Grid

Connect Infill Lines

Infill Line Directions []

Infill Line Multiplier 1

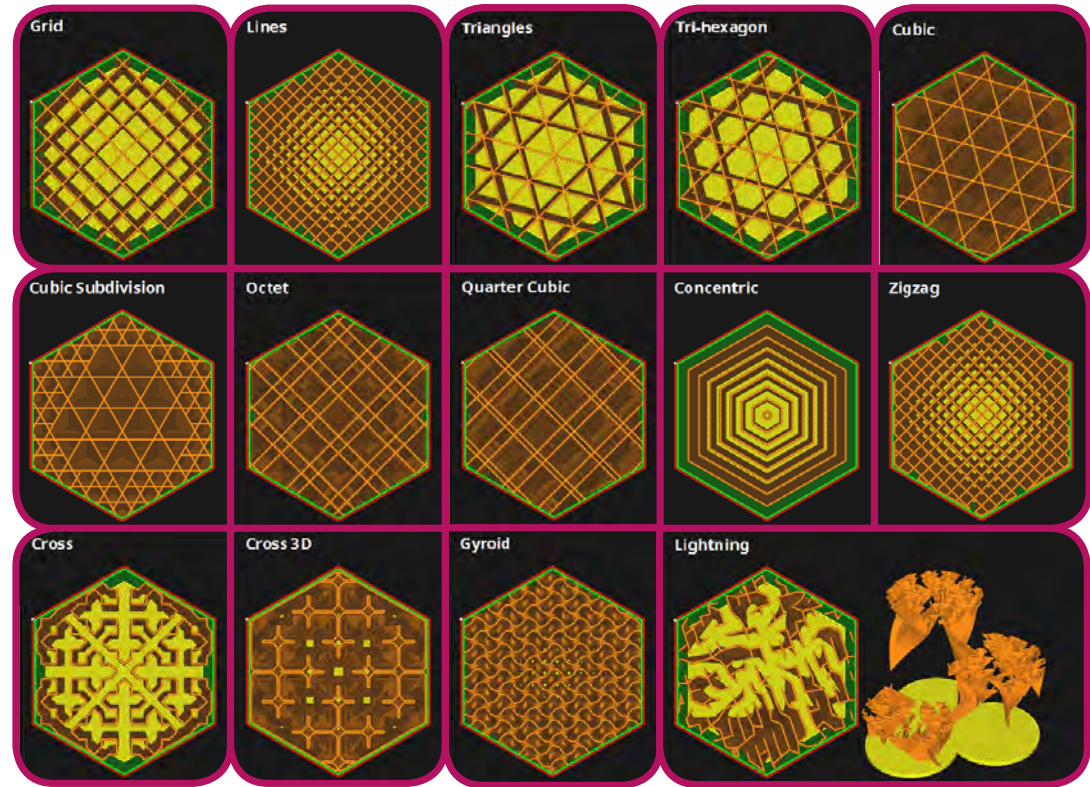
Infill Overlap Percentage 10.0 %

Infill Layer Thickness 0.15 mm

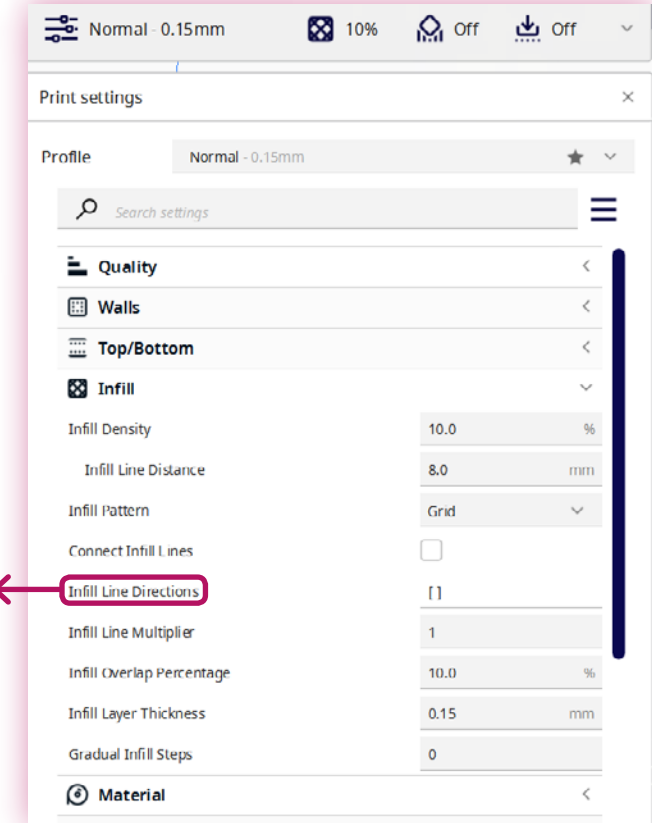
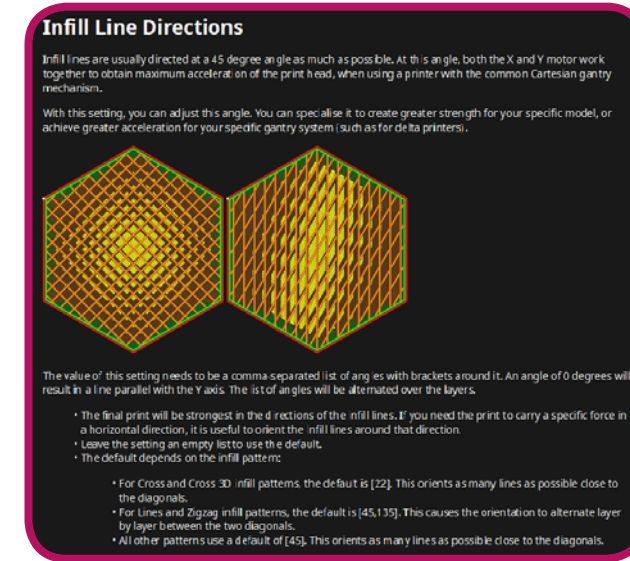
Gradual Infill Steps 0

Material

INFILL - PATTERN



INFILL - LINE DIRECTIONS



MATERIAL - PRINTING TEMP.

078

Printing Temperature

This refers to the temperature of the nozzle while printing. The printing temperature is one of the most influential settings, because it affects how the material behaves during printing. Even a small difference in temperature can have a large effect on how the plastic flows.

Every type of filament has a range of temperatures it can be printed at. This will usually be listed on the box it came in and on the technical data sheet. Increasing the temperature generally makes thermoplastics more fluid. This allows the printer to extrude material faster, because the internal friction is lowered. When printing with large layer heights, wide lines, large flow or great speeds, the temperature needs to be towards the higher end of the temperature range. However, hotter printing also makes the material harder to cool, so this causes overhangs to sag more (requiring more support) and causes more stringing.

Setting the temperature too high causes the material to degrade while printing. This can clog your nozzle and potentially damage your printer. Setting the temperature too low causes the feeder to grind on the material, making it stop extruding.

Affects

- Printing Temperature Initial Layer
- Initial Printing Temperature
- Final Printing Temperature
- Break Preparation Temperature

Affect ed By

- Default Printing Temperature

Normal - 0.15mm10%OffOff

Print settings

ProfileNormal - 0.15mm

Search settings

QualityWallsTop/BottomInfillMaterial

Printing Temperature205.0°C

Printing Temperature Initial Layer205.0°C

Build Plate Temperature60°C

Build Plate Temperature Initial Layer60°C

Flow100.0%

Wall Flow100.0%

Top/Bottom Flow100.0%

Infill Flow100.0%

Skirt/Brim Flow100.0%

MATERIAL - BUILD PLATE TEMP.

079

Build Plate Temperature

Some printers have a heated build plate. This setting determines how warm the build plate will become.

Heating the build plate will keep the material slightly fluid and sticky. Some materials will form crystals upon freezing, which causes the material to shrink significantly upon a freezing. The heated bed keeps the material just above freezing temperature to prevent this shrinkage and to maintain the stickiness of the liquid plastic. This is all meant to improve the adhesion of the print to the build plate.

However if the build plate is kept too hot, the print will be very fluid where it touches the build plate. This causes the material to sag a bit, causing warping when the higher up material starts to shrink.

If this setting is set to 0 degrees, Cura will not output any commands to change the build plate's temperature, which could confuse the firmware if there is no build plate.

When you adjust the build plate temperature in a material profile, this will adjust the Default Build Plate Temperature setting. Normally the Build Plate Temperature will be equal to the Default Build Plate temperature, but sometimes choosing a different quality can make slight adjustments to the build plate temperature. This Build Plate Temperature setting is the one that actually gets used for the print.

Affects

- Build Plate Temperature Initial Layer

Affect ed By

- Default Build Plate Temperature

Normal - 0.15mm10%OffOff

Print settings

ProfileNormal - 0.15mm

Search settings

QualityWallsTop/BottomInfillMaterial

Printing Temperature205.0°C

Printing Temperature Initial Layer205.0°C

Build Plate Temperature60°C

Build Plate Temperature Initial Layer60°C

Flow100.0%

Wall Flow100.0%

Top/Bottom Flow100.0%

Infill Flow100.0%

Skirt/Brim Flow100.0%

MATERIAL - FLOW

Flow

Adjusting the flow directly adjusts how much material is placed down. Normally the amount of material is calculated by Cura such that the material would fill the exact space in the lines width, height and length, but this can be adjusted using the flow settings.

The main use case for adjusting the flow is to compensate for some fault in the extrusion train. For instance, if the nozzle is slightly clogged, this can cause underextrusion. Increasing the flow could then push more material through the clogged nozzle, hopefully causing it to end at the right amount of material. This can also compensate for slip or variance in filament diameter.

It is almost always better to fix the actual problem causing the overextrusion or underextrusion, but adjusting the flow can be an easy temporary fix.

If you want the lines to be wider, it is better to adjust the actual line width setting. The line width doesn't necessarily need to be the same as the nozzle size. Adjusting the line width will also adjust the spacing between the lines which prevents overextrusion and underextrusion. Adjusting the flow doesn't adjust spacing.

Affects

- Wall Flow
- Top/Bottom Flow
- Infill Flow
- Skirt/Brim Flow
- Support Flow
- Support Interface Flow
- Prime Tower Flow

Normal - 0.15mm 10% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality Walls Top/Bottom Infill Material

Printing Temperature 205.0 °C

Printing Temperature Initial Layer 205.0 °C

Build Plate Temperature 60 °C

Build Plate Temperature Initial Layer 60 °C

100.0 %

Wall Flow 100.0 %

Top/Bottom Flow 100.0 %

Infill Flow 100.0 %

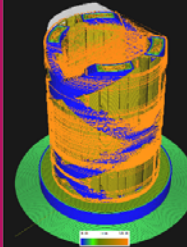
Skirt/Brim Flow 100.0 %

SPEED - PRINT SPEED

Print Speed

This setting adjusts the overall speeds at which the model is printed.

Since speeds can also be adjusted separately for individual structures of the print, the actual printing speed will still vary. This setting causes the rest of the speeds to adjust by ratio.



This setting is a great influence on the balance between printed part quality and printing time. Increasing the printing speed will reduce the duration of the print, but will increase vibration of the print head. It will also make it more difficult for the feeder to keep up, making the print more sensitive to overextrusion and underextrusion.

When increasing the print speed, it is advisable to increase the material temperature as well. This will make the material slightly more runny, which allows it to flow more easily at greater speeds.

Increasing the printing speed will not have a large effect if the print has many small details. Acceleration and jerk are the bottleneck then.

Affects

- Infill Speed
- Wall Speed
- Top/Bottom Speed
- Support Speed
- Prime Tower Speed
- Travel Speed
- Initial Layer Speed
- Initial Layer Travel Speed
- Raft Print Speed
- Maximum Travel Resolution

Normal - 0.15mm 10% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Quality Walls Top/Bottom Infill Material Speed

Print Speed 60.0 mm/s

Infill Speed 100.0 mm/s

Wall Speed 30.0 mm/s

Top/Bottom Speed 30.0 mm/s

Travel Speed 120.0 mm/s

Initial Layer Speed 30.0 mm/s

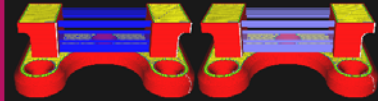
Skirt/Brim Speed 30.0 mm/s

Z Hop Speed 10.0 mm/s

TRAVEL - RETRACTION

Enable Retraction

If a 3D printer would stop pushing material, the material won't immediately stop flowing from the end of the nozzle. There is some residual material in the nozzle and it'll be pressurised, so it keeps flowing for a while longer. To actually stop the material from flowing, the printer needs to retract the material out of the nozzle opening. This is necessary to make clean travel moves without stringing.



Retractions are done when a travel move is made that is particularly sensitive to stringing. Travel moves that only pass through infill or travel from support to support are normally not retracted. There is also a limit to how often the material can be retracted, through the Minimum Extrusion Distance Window settings.

Retracting has significant advantages, but also some disadvantages:

- It'll reduce stringing significantly. No more wiring between different parts of the print.
- It will significantly reduce the amount and size of blobs on the surface where the nozzle enters the perimeter of a part.
- It takes a bit of time to retract.
- The material flow gets interrupted when a retraction happens. This has negative consequences for dimensional accuracy, bending and underextrusion.
- The filament could wear down when too many retractions happen, preventing the feeder from gripping the material.

Flexible materials are harder to retract, because pulling on the filament makes the filament stretch instead of pulling it out of the nozzle. It may be very time consuming and relatively ineffective to enable retraction with such materials.

Affects

- Wipe Retraction Enable

Normal - 0.15mm 10% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Top/Bottom

Infill

Material

Speed

Travel

Enable Retraction ☒

Retract at Layer Change ☐

Retraction Distance 0.8 mm

Retraction Speed 70.0 mm/s

Combing Mode All

Avoid Printed Parts When Traveling ☒

Avoid Supports When Traveling ☐

Travel Avoid Distance 0.625 mm

Z Hop When Retracted ☐

COOLING - PRINT COOLING

Enable Print Cooling

This setting will enable or disable the fans on the print head during printing. The fans are meant to cool down the material while it is printing, so that it solidifies faster.

Materials with a low glass transition temperature range, such as PLA, will need to be cooled down while printing. The fans help to do this by blowing cooler air from the environment onto the hot material that just exited the nozzle. Otherwise it will start to sag from the heat, which causes deformities and may even fail the print completely. In places where the material is hanging in mid-air, such as where there is an overhang, there is nothing to stop the sagging so there it is critical that the material is cooled immediately.

For materials with a higher glass transition temperature range, such as ABS, it is still recommended to turn on the fans, but they may spin at a lower rate for certain parts of the print. Most printers allow precise control over the fan speed, so it's not just an on/off toggle. You can usually control the precise fan speed.

Only for materials with a very high glass transition temperature should you completely disable the fans. If the fans are enabled then, it could give extrusion problems and make the final print brittle.

Affects

- Fan Speed

Normal - 0.15mm 10% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

Walls

Top/Bottom

Infill

Material

Speed

Travel

Cooling

Enable Print Cooling ☒

Fan Speed 100.0 %

Initial Fan Speed 0.0 %

Regular Fan Speed at Height 0.2 mm

Regular Fan Speed at Layer 2

Support

Build Plate Adhesion

SUPPORT - GENERATE SUPPORT

Generate Support

3D printers with fused filament fabrication work by placing a string of molten plastic in the correct locations. If this plastic is not supported, it will sag down, because it gets pushed out the nozzle with significant force and because of gravity. Unless the model is beneath it to support the new layer, it will need to be supported by an additional support structure that is removed and discarded after printing the model.

This setting enables creating support structures to support the object while it's being printed. Cura won't enable this support by default, but it will mark the surface of the model red if it thinks that the material there could sag a lot if support is not enabled.



Printing support takes a significant amount of time and material. Where the support touches the model it needs to be broken or cut off later, after the print has completed. This tends to leave a scar on the surface.

However support is sometimes completely necessary. For instance, if the model has a piece that points down towards the build plate, this piece would otherwise just hang in mid-air.

Design for preventing support

While it is sometimes necessary, printing support should be prevented if at all possible. There are numerous techniques to prevent having to print support. This is a large part of model design for 3D printing. Here are a couple of techniques by which your 3D model can be altered to prevent needing support.

- Orient your model such that there are no flat surfaces higher than the build plate.
- Avoid designing overhangs as much as possible.
- If there is a surface just above your build plate, pull it down to the build plate.
- Small ledges can be supported with a 45 degree angle slope, transferring the overhang into the model.
- Larger overhangs can be bridged if they are supported on both sides. Cura will automatically orient the lines of the bottom side of that overhang such that they go in a straight line over the bridge. During the printing of these lines, the nozzle will keep tension on the bead such that it doesn't droop down. This then gives it enough time to solidify so that it can hold up on its own.
- If there is a gap in the overhang, a sacrificial bridge can be designed in. This is one layer in which the gap is closed. This allows for good bridging, such that the second layer can build the proper walls of the gap on top. After printing, the gap can be opened with a knife, since it's only one layer thick.

Normal - 0.15mm 10% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

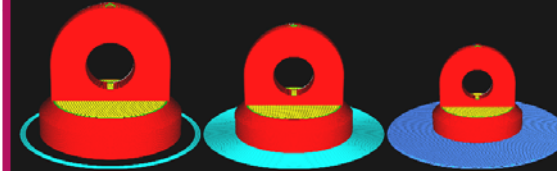
- Quality
- Walls
- Top/Bottom
- Infill
- Material
- Speed
- Travel
- Cooling
- Support
- Build Plate Adhesion
- Experimental

Generate Support

BUILD PLATE ADHEASION

Build Plate Adhesion Types

There are three types of build plate adhesion: skirt, brim and raft. You can also easily disable the adhesion types by setting it to None.



Skirt

A skirt is a single line encircling your print. It will not directly contribute anything to build plate adhesion. However if you don't want to use the other adhesion methods, this method still serves two functions.

- It causes your nozzle to prime before starting the printing of your actual model, to make sure that material is properly flowing.
- It allows you to see whether or not your build plate is properly levelled.

Brim

A brim is a single layer flat area around the base of your model. Its purpose is to keep the edges of your print down and make the contact area between your print and the build plate bigger.

- The bigger surface area allows your print to stick better to the build plate. This is useful for pretty much any print larger than a few centimetres.
- This brim will also keep the edges of your print down. Materials that shrink a lot when cooling down (like ABS) tend to warp a lot while printing. Having a brim with sufficient width can keep the corners in place, preventing this warping.

Raft

A raft is a thick plate between the model and the build plate. This raft shields your print from the heat of the build plate. It will also have a lot of surface area to stick to the build plate with. The print gets put on top of this raft, where it may stick much better. Be careful with thin prints though, since it may be hard to remove the raft without damaging the print.

Affects

- Regular Fan Speed at Height
- Prime Tower X Position
- Prime Tower Y Position

Normal - 0.15mm 10% Off Off

Print settings

Profile Normal - 0.15mm

Search settings

- Quality
- Walls
- Top/Bottom
- Infill
- Material
- Speed
- Travel
- Cooling
- Support
- Build Plate Adhesion
- Experimental

Build Plate Adhesion Type Skirt

TABLE OF FIGUES

Figure/ Abbildung	page/ Seite	description/ Beschreibung	source/ Quelle
coverpicture/ Titelbild	Cover/ Deckblatt	detail 3D printed structure/ Detail 3D Druckstruktur	own photography/ eigene Fotografie
left/links	004	rendering of 3D print sketch/ Rendering einer 3D-Druck Illustration	own rendering created in Rhinoceros 3D/ eigenes Rendering erstellt in Software Rhinoceros 3D
top left/ oben links	008	technical drawing von Stratasys Inc/ echnische Zeichnung	Swanson, W.J. et al. (2004) 'High temp- erature modeling apparatus'. Available at: https://patents.google.com/ patent/US6722872B1/en (Accessed: 6 October 2023).
bottom left/ unten links	008	logo Shenzhen Creality 3D Technology Co., Ltd	creality (2022). Available at: https://www.creality.com/pages/about-us (Accessed: 6 October 2023).
top right/ oben rechts	009	logo Stratasys Inc	Stratasys (2022). Available at: https://www. stratasys.com/de/ (Accessed:9October2023).
bottom right/unten rechts	009	logo UltiMaker	UltiMaker (2011) UltiMaker. Available at: https://ultimaker.com/ (Accessed: 6 October 2023).

ABBILDUNGSVERZEICHNIS

Figure/ Abbildung	page/ Seite	description/ Beschreibung	source/ Quelle
top left/ oben links	010	carbon fiber reinforced 3D printed parts/ kohlefaserverstärkte 3D-Druckteile	Mark3D, T. (2016) 'Der Mark Two 3D-Drucker von Markforged', Mark3D Markforged 3D Drucker. Available at: https://www.mark3d.com/de/markforged- mark-two/ (Accessed: 6 October 2023).
bottom left/ unten links	010	logo Mark3D GmbH	Mark3D, T. (2016) 'Der Mark Two 3D-Drucker von Markforged', Mark3D Markforged 3D Drucker. Available at: https://www.mark3d.com/de/markforged- mark-two/ (Accessed: 6 October 2023).
top right/ oben rechts	011	logo BigRep GmbH	'BigRep GmbH Large-Scale 3D Printers Manufacturing Company' (2018), 6 August. Available at: https://bigrep.com/company/ (Accessed: 6 October 2023).
bottom right/unten rechts	011	example for 3D printed furniture/ Beispiel für 3D-gedruckte Möbel, Bow chair (left/ links) & Rise Chair (right/ rechts) by Nagami x Zaha Hadid Architects	'Zaha Hadid Architects' (2018). Available at: https://www.zaha-hadid.com/design/rise- chair/ (Accessed: 29 October 2023).

TABLE OF FIGUES

Figure/ Abbildung	page/ Seite	description/ Beschreibung	source/ Quelle
top left/ oben links	012	logo Prusa Research	Prusa Research (2022) Prusa3D by Josef Prusa. Available at: https://www.prusa3d.com/de/page/uber-uns_77/?nextRoute=page (Accessed: 6 October 2023).
bottom left/ unten links	012	3D printer Prusa i3 MK3S+	own photography/ eigene Fotografie
top right/ oben rechts	013	logo Bambu Lab	Bambu Lab (2023). Available at: https://bambulab.com/de-de/about-us (Accessed: 6 October 2023).
bottom right/unten rechts	013	picture of Creality Ender 3 V2	own photography/ eigene Fotografie
1. position left/ links	014	logo Blender	Blender project (2023). Available at: https://www.blender.org/ (Accessed: 29 November 2023).
2. position left/ links	014	logo FreeCAD	FreeCAD (2023). Available at: https://www.freecad.org/index.php?lang=de (Accessed: 29 November 2023).
3. position left/ links	014	logo SketchUp Make	SketchUp (2023). Available at: https://www.sketchup.com/ (Accessed: 29 November 2023).
4. position left/ links	014	logo Rhinoceros 3D	McNeel, R. (1993 - 2023) Rhinoceros 3D. Available at: https://www.rhino3d.com/ (Accessed: 29 November 2023).

ABBILDUNGSVERZEICHNIS

Figure/ Abbildung	page/ Seite	description/ Beschreibung	source/ Quelle
5. position left/ links	014	logo Fusion 360	Autodesk Fusion 360 (2023). Available at: https://www.autodesk.de/products/fusion-360/overview (Accessed: 29 November 2023).
1. position left/ links	015	logo UltiMaker Cura	UltiMaker (2011) UltiMaker. Available at: https://ultimaker.com/ (Accessed: 6 October 2023).
2. position left/ links	015	logo Slic3r	Slic3r (2023). Available at: https://slic3r.org/ (Accessed: 29 November 2023).
3. position left/ links	015	logo PrusaSlicer	Prusa Research (2022) Prusa3D by Josef Prusa. Available at: https://www.prusa3d.com/de/page/uber-uns_77/?nextRoute=page (Accessed: 6 October 2023).
4. position left/ links	015	logo Creality Slicing Software	creality (2022). Available at: https://www.creality.com/pages/about-us (Accessed: 6 October 2023).
5. position left/ links	015	logo Simplify3D	'Simplify3D Software' (2023). Available at: https://www.simplify3d.com/ (Accessed: 29 November 2023).

TABLE OF FIGUES

Figure/ Abbildung	page/ Seite	description/ Beschreibung	source/ Quelle
bottom right/unten rechts	015	Screenshot from Software Ultimaker Cura with Plug-In 'Settings Guide'/ Screenshot aus Software Ultimaker Cura mit Plug-In 'Settings Guide'	Settings Guide - Ultimaker Cura Market-place (2022). Available at: https://market-place.ultimaker.com/app/cura/plugins/Ghostkeeper/SettingsGuide2 (Accessed: 29 November 2023).
left and right/ links und rechts	016/017	rendering of 3D model/ Rendering eines 3D-Modells	own rendering created in Rhinoceros 3D/ eigenes Rendering erstellt in Software Rhinoceros 3D
left and right/ links und rechts	018/019	Screenshot from sliced 3D model in Software Ultimaker Cura/ Screenshot von geslicetem 3D Modell in Software Ultimaker Cura	screenshot from Ultimaker Cura/ Screenshot aus Ultimaker Cura
5 figures from left to right/ 5 Abbildungen von links nach rechts	020, 021, 025, 029	rendering of Direct Drive System, Bowden System, FFF 3D printer, print head, printed object/ Rendering eines Direct Drive Systems, Bowden Systems, FFF 3D Druckers, Druckkopfes, Druckobjektes	own rendering created in Rhinoceros 3D/ eigenes Rendering erstellt in Software Rhinoceros 3D
9 figures/ 9 Abbildungen	022, 024, 026, 027, 028, 030	3D printer, two extruders, heat break, heat sink, heating block, nozzle, Close up 3D print / 3D Drucker, zwei Extruder, Halsdüse, Kühlkörper, Heizblock, Druckdüse, Close up 3D Druck	own photography/ eigene Fotografie

ABBILDUNGSVERZEICHNIS

Figure/ Abbildung	page/ Seite	description/ Beschreibung	source/ Quelle
right/rechts	031	Microscope image of print paths (PLA) / Mikroskopaufnahme Druckpfade (PLA)	own microscopy/ eigene Mikroskopie
4 figures/ 4 Abbildungen	032/033, 034, 037	3D print, filament diameter, cabinet full of filaments/ 3D Druck, Filamentdurchmesser, Schrank voller Filamente	own photography/ eigene Fotografie
5 figures/ 5 Abbildungen	039-043	CAD model to STL/ vom CAD-Modell zum STL	screenshot from Rhinoceros 3D/ Screenshot aus Rhinoceros 3D
7 figures/ 7 Abbildungen	044-051	STL to G-Code / vom STL zum G-Code	screenshot from Ultimaker Cura/ Screenshot aus Ultimaker Cura
34 figures/ 34 Abbildungen	052-065	Printer Setup - Post Processing/ Drucker vorbereiten - Nachbearbeitung	own photography/ eigene Fotografie
left/links	066	Slicer Print Settings/ Druckeinstellungen	screenshot from Ultimaker Cura/ Screenshot aus Ultimaker Cura
18 figures/ 18 Abbildungen	068-085	Layerheight - Build Plate Adheasion	Screenshot from Software Ultimaker Cura with Plug-In 'Settings Guide'/ Screenshot aus Software Ultimaker Cura mit Plug-In 'Settings Guide'
backcover/ Rückumschlag	back/ Rückseite	detail 3D printed structure/ Detail 3D Druckstruktur	own photography/ eigene Fotografie

LIST OF REFERENCES

Aerospace 3D printing | Orion Additive Manufacturing Orion Additive Manufacturing GmbH.
Available at: <http://127.0.0.1:8069/aerospace> (Accessed: 9 October 2023).

Ahlers, D. (2018) 3D Printing of Nonplanar Layers for Smooth Surface Generation.

Ariffin, M.K.A.M. et al. (2018) 'Slicer Method Comparison Using Open-source 3D Printer', IOP Conference Series: Earth and Environmental Science, 114(1), p. 012018.
Available at: <https://doi.org/10.1088/1755-1315/114/1/012018>.

Bambu Lab (2023). Available at: <https://bambulab.com/de-de/about-us>
(Accessed: 6 October 2023).

Berman, B. (2012) '3-D printing: The new industrial revolution', Business Horizons, 55(2), pp. 155–162.
Available at: <https://doi.org/10.1016/j.bushor.2011.11.003>.

'BigRep GmbH | Large-Scale 3D Printers Manufacturing Company' (2018), 6 August.
Available at: <https://bigrep.com/company/> (Accessed: 6 October 2023).

Chapman, A. (2022) The complete history of 3D printing, UltiMaker. Available at: <https://ultimaker.com/learn/the-complete-history-of-3d-printing/>
(Accessed: 6 October 2023).

LITERATURVERZEICHNIS

creality (2022). Available at: <https://www.creality.com/pages/about-us>
(Accessed: 6 October 2023).

Dönitz, A. et al. (2023) 'Additive Manufacturing of Biodegradable Hemp-Reinforced Polybutylene Succinate (PBS) and Its Mechanical Characterization', Polymers, 15(10), p. 2271.
Available at: <https://doi.org/10.3390/polym15102271>.

Fastermann, P. (2016) '3D-CAD-Zeichensoftware und Umgang mit 3D-Druck-Daten', in P. Fastermann (ed.) 3D-Drucken: Wie die generative Fertigungstechnik funktioniert. Berlin, Heidelberg: Springer (Technik im Fokus), pp. 49–54.
Available at: https://doi.org/10.1007/978-3-662-49866-8_6.

Gebhardt, A. (2013) 'Einordnung und Begriffsbestimmung', in Generative Fertigungsverfahren. Carl Hanser Verlag GmbH & Co. KG, pp. 1–19.
Available at: <https://doi.org/10.3139/9783446436527.001>.

Gibson, I. et al. (2021) 'Binder Jetting', in I. Gibson et al. (eds) Additive Manufacturing Technologies. Cham: Springer International Publishing, pp. 237–252.
Available at: https://doi.org/10.1007/978-3-030-56127-7_8.

Gieseke, M. et al. (2016) 'Laserbasierte Technologien', in R. Lachmayer, R. bastian Lippert, and T. Fahlbusch (eds) 3D-Druck beleuchtet: Additive Manufacturing auf dem Weg in die Anwendung. Berlin, Heidelberg: Springer, pp. 19–30.
Available at: https://doi.org/10.1007/978-3-662-49056-3_3.

LIST OF REFERENCES

Grasshopper Kangaroo Tutorial (Soap Film) (2020). Available at: <https://www.youtube.com/watch?v=-VagvK9CIK4>

(Accessed: 23 October 2023).

Grothe, T., Brockhagen, B. and Storck, J.L. (2020) 'Three-dimensional printing resin on different textile substrates using stereolithography: A proof of concept', Journal of Engineered Fibers and Fabrics, 15, p. 1558925020933440. Available at: <https://doi.org/10.1177/1558925020933440>.

Koch, H.C., Schmelzeisen, D. and Gries, T. (2021) '4D Textiles Made by Additive Manufacturing on Pre-Stressed Textiles—An Overview', Actuators, 10(2), p. 31.

Available at: <https://doi.org/10.3390/act10020031>.

Ligon, S.C. et al. (2017) 'Polymers for 3D Printing and Customized Additive Manufacturing', Chemical Reviews, 117(15), pp. 10212–10290.

Available at: <https://doi.org/10.1021/acs.chemrev.7b00074>.

Mark3D, T. (2016) 'Der Mark Two 3D-Drucker von Markforged', Mark3D Markforged 3D Drucker. Available at: <https://www.mark3d.com/de/markforged-mark-two/>

(Accessed: 6 October 2023).

McNeel Wiki - Lightweight Extrusion Objects (2020). Available at: https://wiki.mcneel.com/rhino/rhinov5status_extrusions

(Accessed: 23 October 2023).

Prusa Research (2022) Prusa3D by Josef Prusa. Available at: https://www.prusa3d.com/de/page/uber-uns_77/?nextRoute=page

(Accessed: 6 October 2023).

LITERATURVERZEICHNIS

Sitotaw, D.B. et al. (2020) 'Additive Manufacturing and Textiles—State-of-the-Art', Applied Sciences, 10(15), p. 5033.

Available at: <https://doi.org/10.3390/app10155033>.

Stephens, B. et al. (2013) 'Ultrafine particle emissions from desktop 3D printers', Atmospheric Environment, 79, pp. 334–339.

Available at: <https://doi.org/10.1016/j.atmosenv.2013.06.050>.

Stratasys (2022). Available at: <https://www.stratasys.com/de/>

(Accessed: 9 October 2023).

Svetlizky, D. et al. (2021) 'Directed energy deposition (DED) additive manufacturing: Physical characteristics, defects, challenges and applications', Materials Today, 49, pp. 271–295. Available at: <https://doi.org/10.1016/j.mattod.2021.03.020>.

Swanson, W.J. et al. (2004) 'High temperature modeling apparatus'. Available at: <https://patents.google.com/patent/US6722872B1/en>

(Accessed: 6 October 2023).

The Mark Two (2023). Available at: <https://markforged.com/de/resources/blog/the-mark-two> (Accessed: 29 October 2023).

UltiMaker (2011) UltiMaker. Available at: <https://ultimaker.com/> (Accessed: 6 October 2023).

'Zaha Hadid Architects' (2018). Available at: <https://www.zaha-hadid.com/design/rise-chair/> (Accessed: 29 October 2023).

3D PRINTING GUIDE

Antonia Gwendolyn Dönitz

1st part MA thesis

textile & material design

kunsthochschule berlin weißensee

Berlin, Germany

summer 2024

