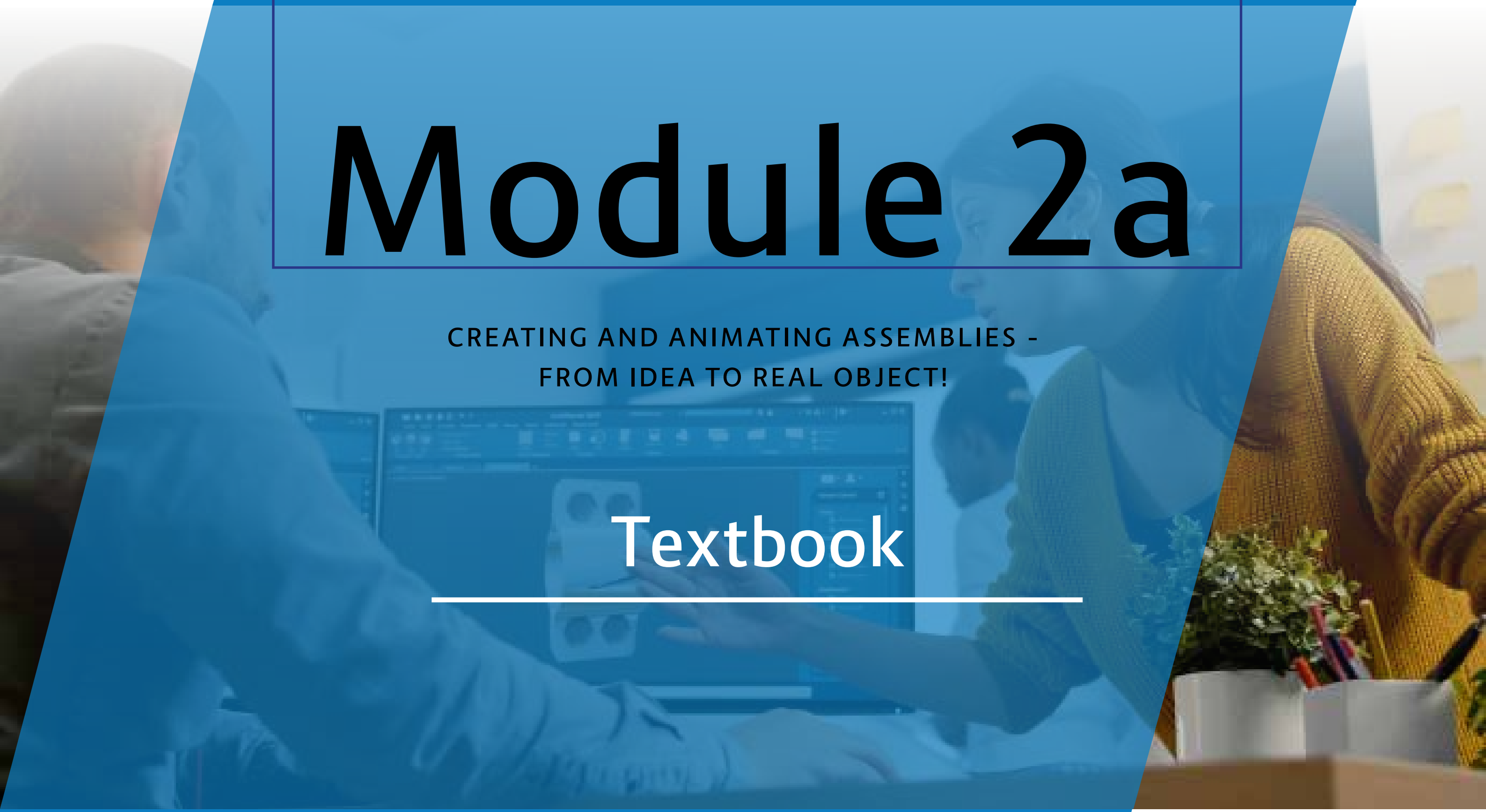
ÜBZO - INTER-COMPANY TRAINING CENTER IN EASTERN BAVARIA 2024

Module 2a

CREATING AND ANIMATING ASSEMBLIES -
FROM IDEA TO REAL OBJECT!



Textbook

Education is Future

MODULAR TRAINING & EDUCATION

IN MECHANICAL
& ELECTRICAL ENGINEERING

TRAINME2

Modular training and further education of South African TVET lecturers in mechanical and electrical engineering

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This textbook has been developed within the framework of the TRAINME & TRAINME 2 programme (Modular Training of South African TVET Lecturers in Mechanical and Electrical Engineering). The TRAINME programmes have been designed by the Inter-Company Training Centre in Eastern Bavaria (ÜBZO) and the University of Stuttgart (Dept. BPT) on behalf of the German Federal Ministry of Education and Research (BMBF) and the DLR Project Management Agency (DLR-PT) in cooperation with the South African Department of Higher Education (DHET).

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Preamble

Welcome to „Creating and Animating Assemblies - From Idea to Real Object“!

This book has been specifically designed for mechanical teachers at TVET colleges who want to prepare their students for the technological future in both practice and theory. Our vision is to take you on an inspiring journey of designing your own teaching materials and turning them into physical objects using a 3D printer.

The world of modern design and technology is constantly changing, and we strongly believe that it is vital that our students are prepared for the challenges of the industry of the future. With the advent of 3D printing, we have the opportunity to turn our ideas and designs into stunning reality and prepare our students for this exciting technological revolution.

This book is a comprehensive guide to CAD (Computer-Aided Design) assembly creation and animation using moving parts. Our aim is to help you develop complex designs and bring your ideas to life through animation. As a result, you will be able to engage your students with attractive and interactive teaching materials that help them to connect with the subject matter.

There will also be a focus on 3D printing, which opens up a whole new dimension to learning and design. We will introduce you to the world of 3D printing with PLA, an affordable and easy-to-use material that allows you to physically print your designs and hand them out to your students.

We also turn our attention to the exciting field of 3D printing with resin, which offers unsurpassed precision and detailed models. With a comprehensive chapter on resin printing, we aim to teach you the basics of this advanced technology and help you create high-quality objects that meet even the most demanding requirements.

Our aim is to empower you as a lecturer to revolutionise teaching by designing your own teaching material and using 3D printed models. We believe this will not only spark your students‘ interest, but also develop their creative and technical skills to prepare them for the exciting and ever-evolving world of technology.

Thank you for choosing „Creating and Animating Assemblies - From Idea to Real Object“. We are confident that this book will provide you with valuable insights and knowledge that you can use in your classroom and take your students on an exciting journey into the technological future.

The author

Content

Unit 1

Why do we create assemblies?

Have you ever wondered how the assemblies on the loading screen were created, or what they do other than advertise the CAD (Computer-Aided Design) programme?

Were you, like me, put off at first and worried whether you could cope with the complexity? In my case I was. Designing individual parts was the first challenge for me. But as time went on, I had to rise to the challenge and focus on the assemblies. I wanted to see if my ideas were feasible and what needed to be done to the design to make it fit together. And as it happened, some things didn‘t fit together the way I had imagined them in my head. So 1:0 for the CAD programme.

But how could I modify the components so that they would fit together on one side and not interfere with the function on the other? The next panic attack came when I imagined having to reassemble the components each time. Problem after problem pushed my motivation to create an assembly further and further away. Of course, you only face the problems when they are there. Fortunately, many people before me had the same problems and published their experiences and, to my delight, their solutions on various websites. And what I learned from some of the help points made it easier for me to create assemblies. The most basic insight was that you only need to adapt the relevant components and then update the assembly. This made it possible to adapt the assemblies to my ideas at any time without having to redesign the design from scratch. For this time saving I would gladly accept a 2:0 in favour of a CAD programme.

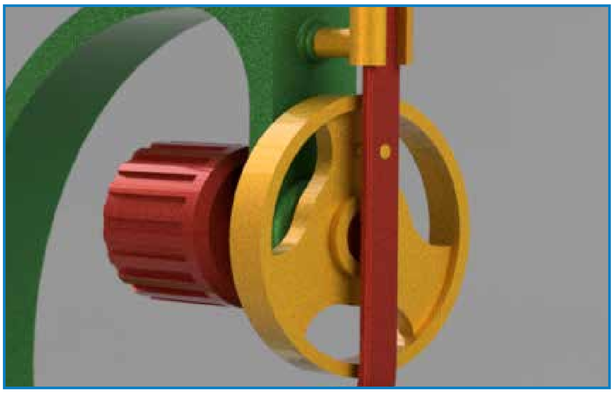
With the relief of overcoming this hurdle, the next step was to question the function of the assembly. There were two approaches to choose from. The first was my tried and tested method of turning, milling, drilling, 3D printing and then assembling the parts. If something doesn‘t work, I can tweak it to make it work. That way has worked in the past. If you ignore the qualitative approach, you can talk yourself into it. As the saying goes, „hope dies last“. Up to this point I had no personal experience of the second option. What would I learn by adding motion links and simulating the processes? Could this approach help me identify errors earlier and find solutions in advance?

There are many examples on the internet of how motion links and their parameterisation can be used. After using these and the subsequent simulation, it quickly became clear that the assembly could not work.

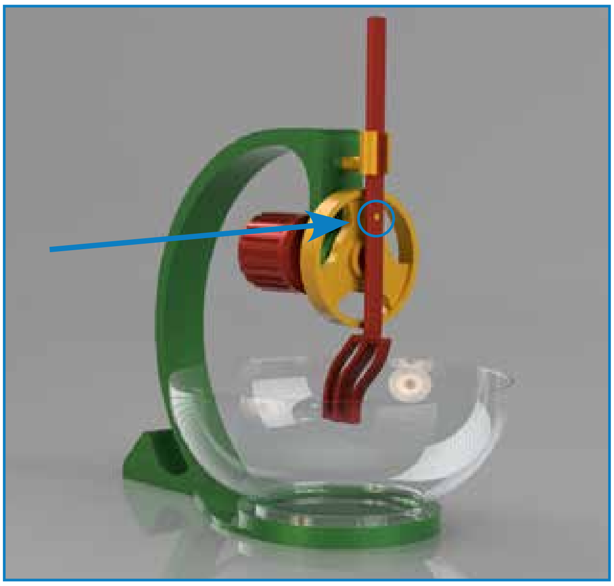
Problems with the mixing bowl

Due to the rotational movement, the linear movement was too far down, which would have damaged the mixing bowl. It was therefore necessary to limit the stroke so that the mixing bowl would not be damaged, but at the same time ensure that the medium to be mixed could still be stirred at the bottom of the bowl.

This movement restriction was adjusted by changing the pin on the turntable downwards as well as the hole in the movement arm in the respective design. When the assembly was updated, the changes made were adopted. It was then possible to run another simulation to check the changes.



After the first assemblies I created, I could see many benefits for myself, which made me think about how my previous way of working on projects was limiting me. Granted, I was always getting a result. But was the quality of my work as high as it could have been? Was I unnecessarily costing my employer material and time that could have been avoided? These are just a few thoughts on the benefits of working with CAD.



1.1 ADVANTAGES AND DISADVANTAGES OF CAD ASSEMBLY GROUPS AND THEIR ANIMATION

Open source programmes are subject to constant change in order to drive continuous improvement of the product. For this reason, the system requirements for Fusion360 are also subject to constant change.

Advantages

- Visualisation: By creating assemblies and animating them, complex mechanical structures and functionalities can be illustrated in a visual way. These allow for better understanding and communication of complicated constructions.
- Error detection: Through animation, potential collision points, movement restrictions or other design errors can be detected and corrected at an early stage, before the product is physically created. This saves time and costs in product development.
- Interactive feedback: Animated assemblies allow engineers and designers to test and optimise the movement and operation of a product before it goes into production. This can improve performance and optimise the user experience.
- Customer presentations: Animated assemblies are valuable tools in customer presentations and marketing activities. They allow customers to see the product in action and better understand how it works.
- Training and education: For teachers and trainers, animated assemblies provide an interactive and lively way to teach complex concepts. Students can better understand and learn the movement and behaviour of machines and products.

Disadvantages

- Time required: The creation of assemblies and animations requires time and resources. Depending on the complexity of the design and animation, this process can be time-consuming.
- Learning curve: Mastering CAD software and animation tools requires training and practice. New users need to invest time to familiarise themselves with the features and capabilities.
- Resource requirements: CAD software and the creation of animated assemblies require powerful computers and graphics cards to run smoothly. These can increase hardware requirements and costs.
- Complexity of assemblies: With very complex assemblies, it can be difficult to represent all the elements and their interactions in the animation, which can lead to limitations in the visual representation.
- Abstraction: Sometimes animated assemblies can appear abstract and may not be as intuitively understood as physical models or prototypes.

Conclusion

The benefits of CAD assemblies and their animation are many, ranging from improved visualisation and understanding of complex structures to early defect detection and product optimisation. Interactive feedback options allow engineers and designers to improve product performance and user experience. In addition, animated assemblies serve as valuable tools for customer presentations and provide educators with an interactive way to communicate complex concepts.

However, the potential drawbacks of CAD assemblies and their animation should not be overlooked. The time and learning curve required to master CAD software and animation tools can be demanding. In addition, creating animated assemblies requires powerful computers and graphics cards, which can increase resource requirements. The complexity of highly detailed assemblies can make them difficult to render in animation, and sometimes animated assemblies can seem abstract and less intuitive than physical models or prototypes.

Overall, however, the advantages of CAD assemblies and their animation outweigh the potential disadvantages, as they help to streamline the development process, save costs, improve product quality and enhance understanding for both customers and students. Careful consideration of the pros and cons, as well as targeted user training, are crucial to realising the full potential of CAD assemblies and their animation in various professional fields.

1.2 WHICH PROFESSIONS USE CAD ASSEMBLIES?

CAD assemblies are used in various professional fields and are an indispensable tool for the efficient design and development of products. Here are some examples of how CAD assemblies are used in different fields:

- **Mechanical engineering:** In mechanical engineering, CAD assemblies are used to design and visualise complex machines and devices. Engineers can analyse the movement and interaction of different components to ensure that the machine functions smoothly and meets requirements.
- **Automotive industry:** In the automotive industry, CAD assemblies are critical to the design of vehicles. From engines and transmissions to chassis and interiors, CAD enables the entire vehicle to be digitally designed and simulated before it is physically built.
- **Architecture:** In architecture, CAD assemblies allow architects to design and visualise complex building structures. They can simulate the movement of doors and windows to check the functionality of a building and optimise the design.
- **Aerospace:** In the aerospace industry, CAD assemblies are used to design aircraft, rockets and satellites. Animation of assemblies helps to analyse aerodynamic properties and optimise flight performance.
- **Electronics:** In the electronics industry, CAD assemblies are used to design circuit boards and electronic components. This makes it possible to design and check complex circuits before they

go into production.

- **Medical technology:** In medical technology, CAD assemblies are used to design medical devices such as prostheses, implants and medical instruments. Animation can help verify the functionality and ergonomics of these devices.
- **Entertainment industry:** In the entertainment industry, CAD assemblies are used for the design of film sets, stages and props. Animation makes it possible to plan and simulate complex motion sequences and special effects.
- **Packaging design:** In packaging design, CAD assemblies can be used to test the functionality of packaging and optimise its design to improve product presentation and protection.

These examples show that CAD assemblies are used in a wide range of professions and make a significant contribution to improving product development, efficiency and creativity in a variety of industries.

Unit 2

The influence of cad in quality management

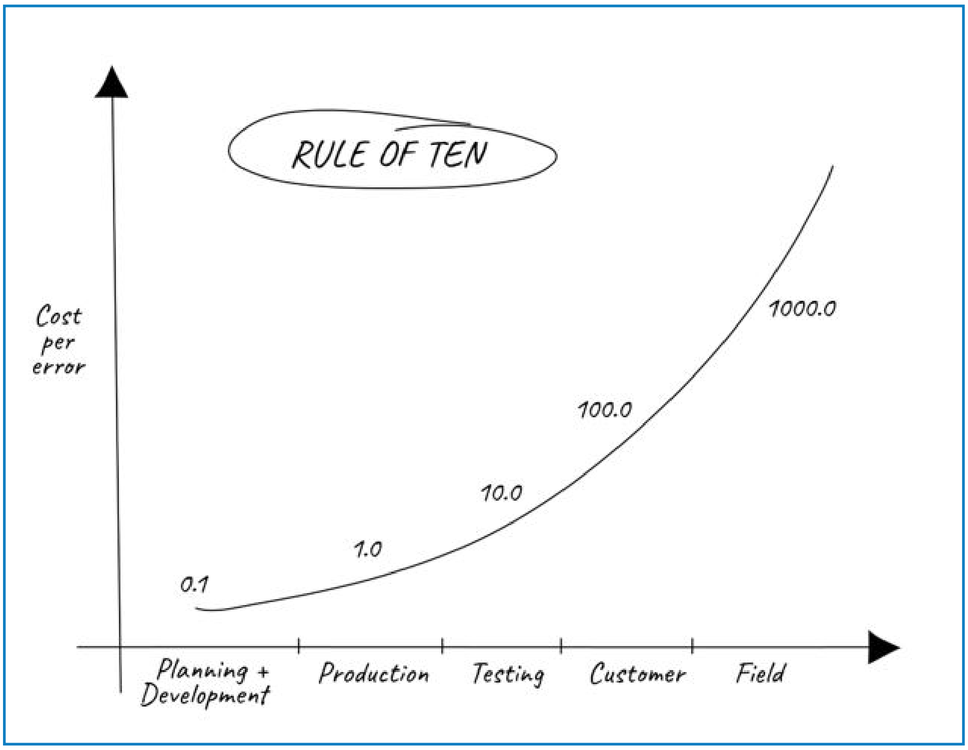
Quality management tools play an important role in the computer-aided design (CAD) environment, helping to improve the quality of products and design processes. Here are some quality management tools that influence the CAD process:

- **Fault analysis and corrective actions:** CAD software often includes fault analysis and corrective action features. By examining models for potential errors or problems, designers can identify and correct errors early, before the product goes into production.
- **Collision and interference checking:** Collision and interference checks can be performed in the CAD environment to ensure that different components or parts do not interfere or collide with each other. These tools help prevent costly errors later in the manufacturing process.
- **Simulation and analysis:** CAD software can be used to perform simulations and analyses to investigate the behaviour of a product under different load conditions. These include finite element analysis (FEA), fluid flow simulation and thermal analysis, which help to evaluate and optimise a product‘s performance and reliability.
- **Design for Manufacturability (DFM):** DFM is an approach that aims to design products that are easier to manufacture and assemble. In the CAD environment, DFM principles can be applied to ensure that the design maximises manufacturing efficiency and minimises manufacturing costs.
- **Geometric Tolerances and Quality Control:** Geometric tolerances can be set in the CAD environment to ensure the precision and quality of products. These tolerances are important to ensure that products meet desired specifications and quality standards.
- **Versioning and Traceability:** CAD software often provides design versioning and change tracking capabilities. This makes it possible to document the development process and track the progress of design changes, which is important to ensure the quality of the final product.

Overall, quality management tools in the CAD environment help to optimise the development process, minimise errors, improve product quality and ensure that the final products meet requirements and standards. By using these tools, companies can work more efficiently, reduce costs and produce higher quality products.

2.1 THE “RULE OF TEN”

When designing assemblies, it is often important to animate their movements in order to eliminate design errors as far as possible in advance. A good illustration of this is the „RULE OF TEN“ in quality management.



The Rule of Ten is a rule of thumb often used in the field of product design and development. It states that the cost and complexity of a product increases by a factor of about 10 as it progresses from an early stage of development (e.g. concept phase) to final series production. This rule does not apply equally to all products, but it still provides useful guidance to emphasise the importance of the early planning stages in a project.

The idea behind the Rule of Ten is that in the early stages of a project it is possible and cost-effective to make changes and design decisions because the impact on overall cost and time is relatively small. In the later stages of a project, on the other hand, changes can be much more expensive and time-consuming because they have already been incorporated into production and may require the use of more expensive materials or special manufacturing processes.

It is therefore advisable to carry out thorough and careful planning and concept development during product development in order to identify and correct potential problems and design errors at an early stage.

By using simulation, prototyping and gathering customer feedback at an early stage, informed decisions can be made to increase the chances of successful and cost-effective product development.

The Rule of Ten emphasises the importance of an interactive and agile development process in which the product is continually improved and adapted before it goes into final production. Early consideration of design aspects, functionality, material selection and production processes can help minimise the impact on overall cost and time and ensure a successful end product.

2.2 WHAT IS THE IMPACT OF CAD ON THE "RULE OF TEN"?

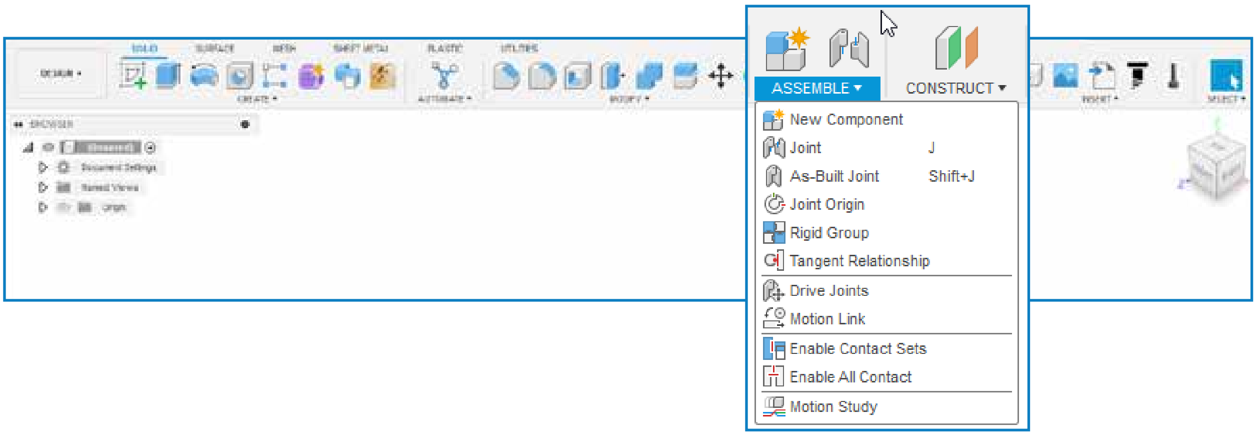
- **Concept and design phase:** In the early stages of a project, it is important to explore and evaluate different concepts and design ideas. CAD makes it possible to quickly and efficiently create different designs, visualise them and analyse their performance. This allows potential problems and challenges to be identified and solutions to be developed before more expensive and time-consuming production begins.
- **Simulation and Analysis:** CAD software provides powerful simulation and analysis tools that allow engineers to study the behaviour of products under different conditions and loads. By running simulations, design flaws, weaknesses or collision points can be identified and fixed early, before the product goes into production.
- **Prototyping:** CAD can be used to create virtual prototypes that closely resemble actual production processes and materials. These prototypes can be used for testing and user feedback to iteratively improve the design and ensure the final product meets the requirements.
- **Material selection and manufacturing optimisation:** CAD allows different materials and manufacturing methods to be tested virtually and their impact on cost, performance and manufacturability to be assessed. This enables engineers to select the best materials and manufacturing processes to minimise the overall cost and complexity of the product.
- **Documentation:** CAD models serve as detailed and accurate documentation of the product design. This information is essential for communicating with the manufacturer, verifying manufacturing standards and producing technical drawings and specifications.

Overall, CAD plays a central role in the Rule of Ten by enabling engineers and designers to comprehensively analyse, simulate and optimise their designs before production begins. By using CAD, design errors can be identified and corrected early on, material selection and production processes can be optimised, and the overall cost and complexity of the product can be minimised. This makes product development more efficient and cost-effective, and increases the chances of success for the final product.

Unit 3

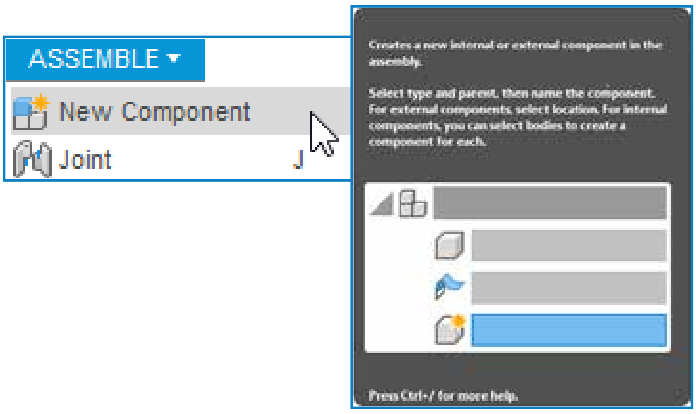
Create assemblies with joints

In the „ASSEMBLE“ section you have the option to combine several components into one assembly. However, it is not possible to combine bodies directly. When you add files, Fusion360 automatically converts them into separate components that can be added to a basic body. Later in the course, this topic will be covered in more detail. First, let‘s take a closer look at the various sub-items.



3.1 NEW COMPONENT

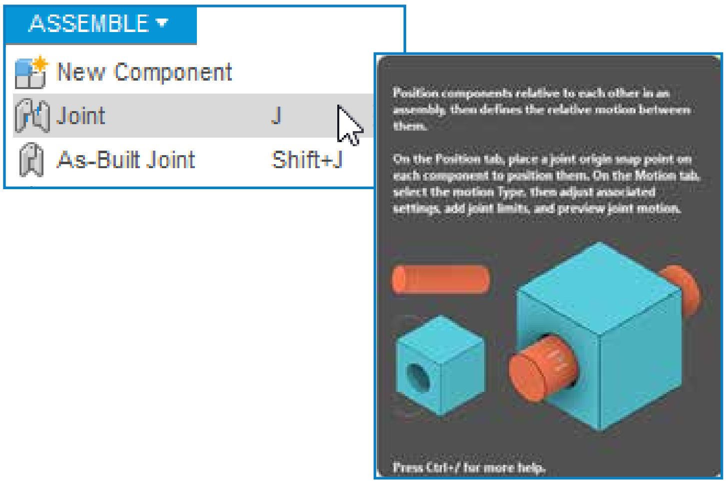
When designing assemblies, it is often important to animate their movements in order to eliminate design errors as far as possible in advance. A good illustration of this is the „RULE OF TEN“ in quality management.



There are several ways to create assemblies. One option is to create the bodies in different folders and then insert them as components into one main body. Another option is to create all the bodies in a single folder and construct each component as a separate body. Use the “New Component” function to convert the required bodies into separate components.

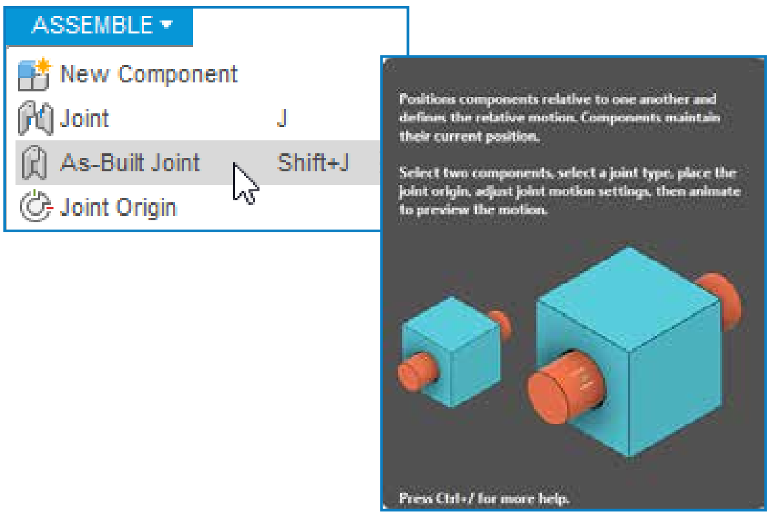
3.2 JOINT

When adding a new component to an existing assembly, it is important to define its position and type of movement. First, drag the component to be added near the assembly and align it accordingly. Then use the „Joint“ command to assign the component a place in the assembly, which you can choose freely. Once the component is in the desired position, set the movement type so that it can move according to the desired requirements.



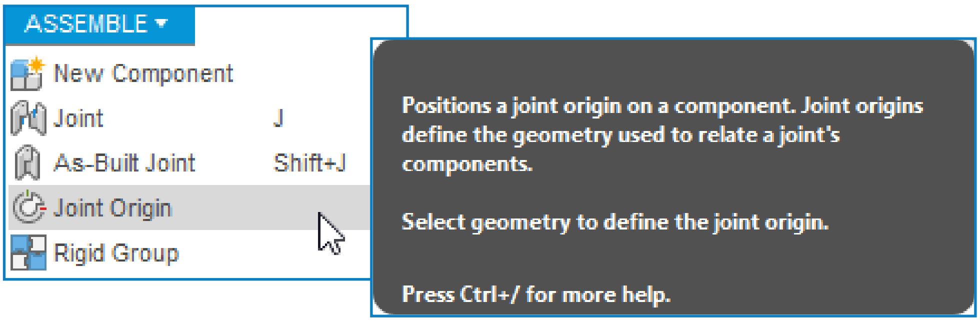
3.3 AS-BUILT JOINT

In the second option, if the created component is already in the desired position relative to another component, the As-Built Joint command can be used to confirm the position. Simply select both components and specify the reference plane for the move type to ensure that the components remain in the desired configuration. This allows you to accurately and efficiently arrange the components in your assembly.



3.4 JOINT ORIGIN

Predefined snap points on the surfaces are used to accurately position the inserted component with an existing one. These are usually at the centre of a surface or at the outer edges. If you wish to create your own snap points, you must define them using Joint Origin. Their position can be set as desired using an X, Y and Z coordinate system.

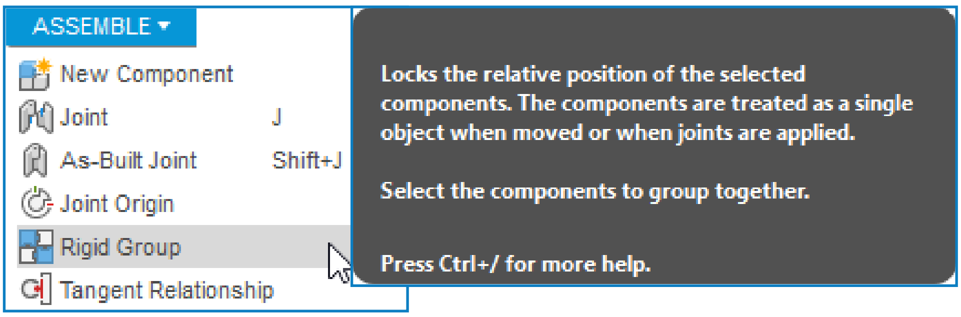


3.5 RIGID GROUP

With the command „Rigid Group" you can lock the possibility of moving at least two components to each other. This state allows you to firmly freeze the moving relationship between the selected components. In other words, the components now behave rigidly to each other and their positions remain unchangeable.

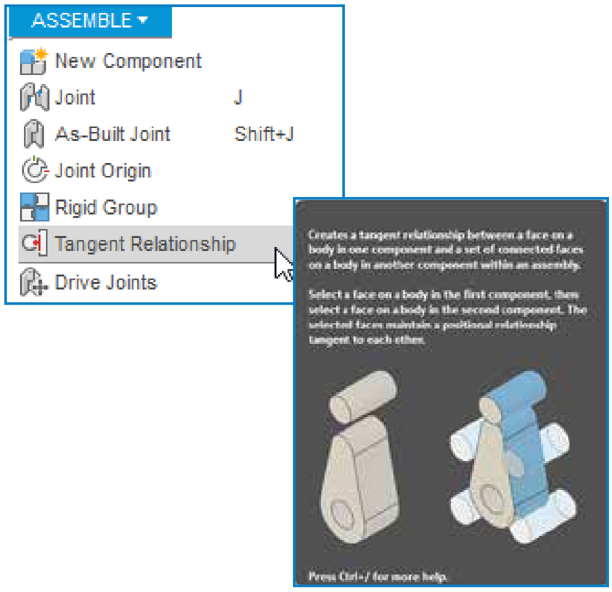
Using the Rigid Group is particularly useful when you want to ensure that certain components in your assembly maintain a fixed position relative to each other and that there is no unintended movement. This ensures the stability and integrity of your design, especially in mechanical applications where a fixed connection is critical.

The ability to lock moving relationships and rigidly group components gives you precise control over the movement and interaction of your assembly. This allows you to create complex designs and ensure that component positions remain exactly as you need them. This feature helps prevent unwanted changes and keeps your design in a defined state.

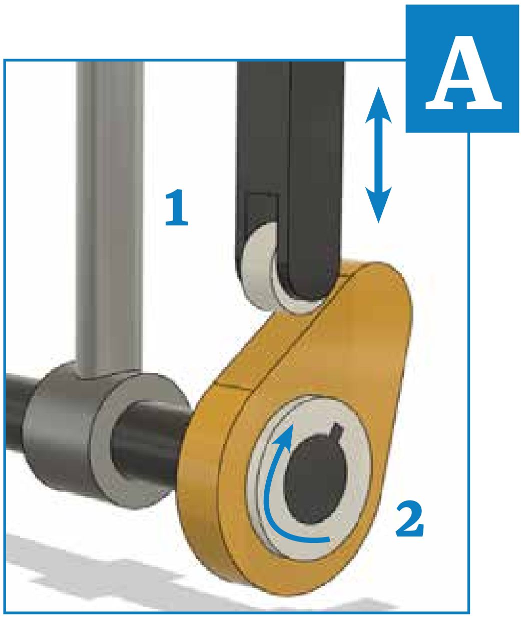


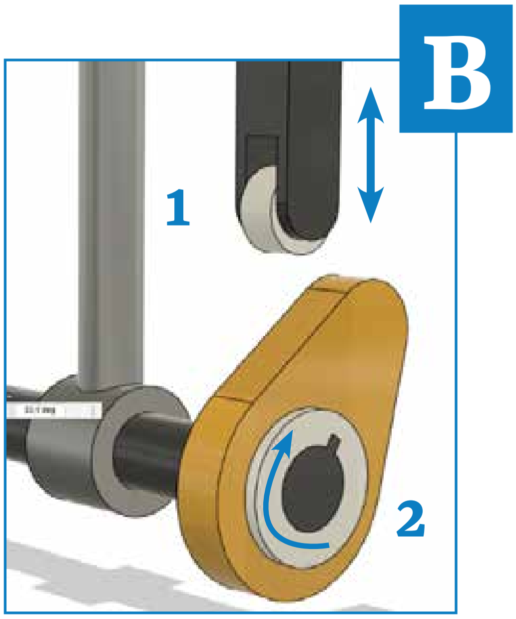
3.6 TANGENT RELATIONSHIP

The command „Tangent Relationship“ allows you to connect two selected surfaces. This dependency makes the two surfaces behave as if they were adjacent to each other, as if they shared a common touch. This creates a seamless connection that makes the geometry of your assembly more precise and realistic.



To illustrate the difference, let‘s consider an example: Imagine that the plunger is to be guided linearly upwards and downwards via a rotating eccentric. The movement type of the linear movement is defined as a „slider“, the rotating one as a „turn”.





In Figure A, in addition to the ‚Slide‘ and ‚Turn‘ motion types, the ‚Tangent Relationship‘ component based constraint has been added. This causes the slide to follow the movements of the eccentric. Figure B shows the difference without the Tangent Relationship. In this case, the plunger is only moved up once and remains in this position, although the eccentric continues to rotate.

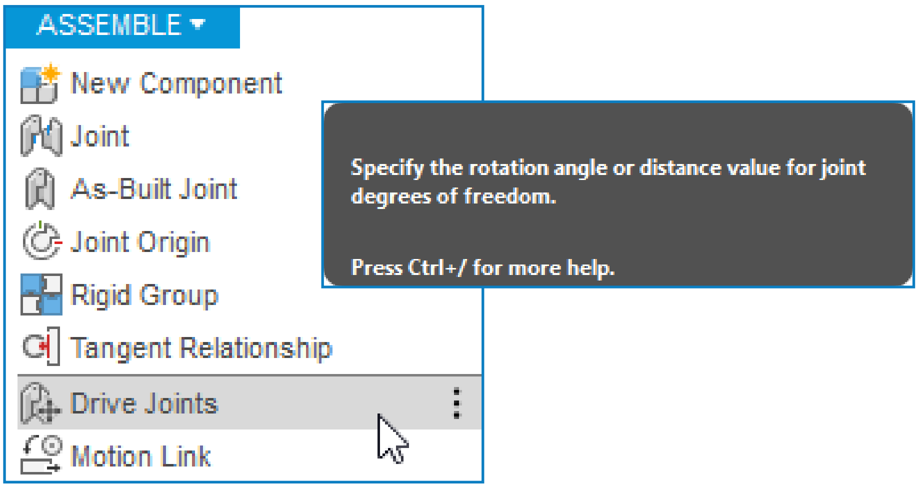
Using the „Tangent Relationship“ allows you to create precise and synchronised movement between components so that they behave realistically as if they were connected. This allows you to perform complex kinematic simulations and ensure that your assembly functions correctly and that all components work together smoothly. This is especially important in engineering and mechanical applications where accurate movement and interaction of components is critical.

3.7 DRIVE JOINT

The Drive Joint command allows you to parameterise movements. For example, you can specify an angle for the home position of a movement type such as „ rotation „. Normally these values are set when a joint is created. The special thing about this is that you can adjust these values at any time, not just in animation mode, but throughout the design process.

The flexibility of the Drive Joint allows you to precisely control the movement of your assembly and simulate different scenarios. You can define parameters that allow you to easily change the motion dynamics and test different positions or states. This is particularly useful when creating interactive designs and exploring multiple configurations of a product.

The ability to parametrically set motion and adjust it at any time gives you complete control over your design and allows you to accurately simulate and optimise complex motion sequences. You can ensure that your product or mechanism works smoothly and efficiently before it goes into production.

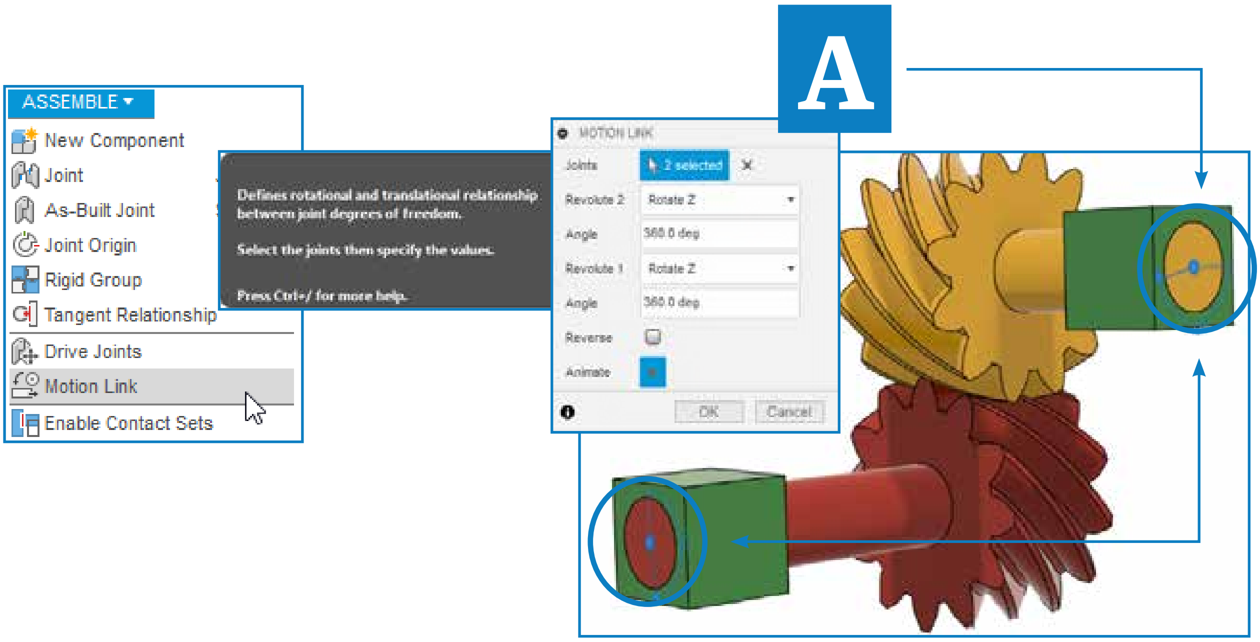


3.8 MOTION LINK

The Motion Link command allows you to make two or more joints dependent on each other to create a linked movement. This allows you to combine joints of the same or different types and precisely control their movement. You can define rotation angles, movement directions and movement lengths as parameters.

We want to link the gears shown so that their motion can be animated synchronously. Select the Motion Link command with a single left click. In the menu window that opens (see figure A), select two pre-existing joints to link them together and create a coordinated motion.

Use "Motion Link" to create complex motion sequences in your assembly and realistically simulate mechanisms. This powerful tool allows you to control the interaction between joints and define precise motion paths. With the ability to combine different joint types, you can create versatile designs and optimise the functionality of your assembly. “Motion Link” provides advanced control over motion in your CAD models, allowing you to design and analyse sophisticated mechanical systems.



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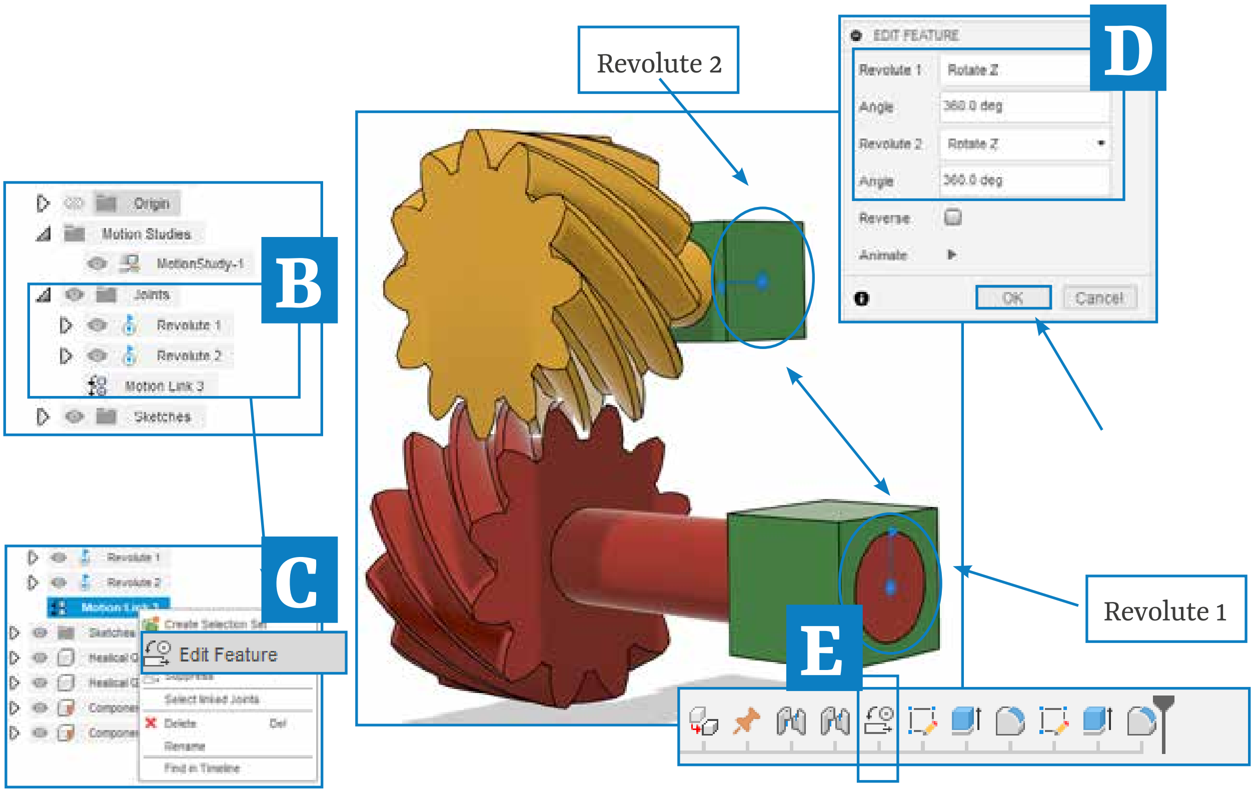


Instead of connecting the gear flanks with a contact (contact set), a Motion Link has been used to avoid high CPU load and to ensure optimal display of the animation.

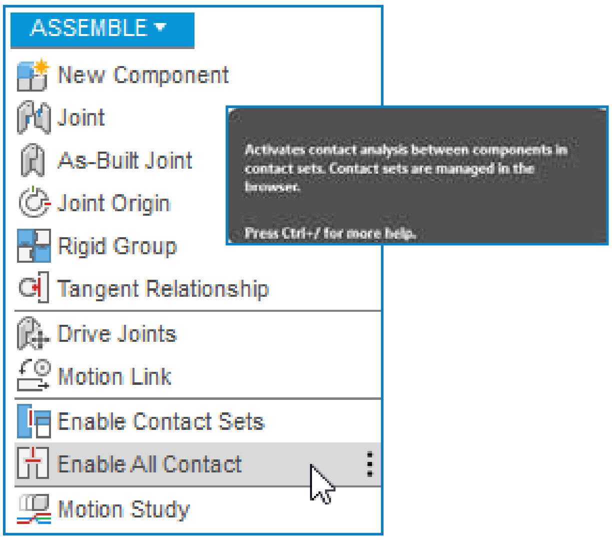
The driving gear „Revolute 1“ animates the driven gear „Revolute 2“ at a ratio of 1:1. A 360° clockwise rotation of the driving gear causes a corresponding 360° clockwise rotation of the driven gear. If the direction of rotation is to be changed to counter-clockwise, the number of degrees is preceded by a minus sign. For example, for a counter-clockwise rotation, the input value would be -360°.

To adjust the values (see figure D), right-click the Motion Link command (see figure B) and select “Edit Feature” from the submenu (see figure C). Change the values as required and confirm with ‚OK‘.

Alternatively, changes can be made using the timeline (see figure E). To do this, double click on the icon and change the values in the additional window (see figure D).

This efficient use of motion links allows you to create complex animations and reduces the CPU load, resulting in a smooth and optimised display of your assembly.

3.9 ENABLE CONTACT SETS



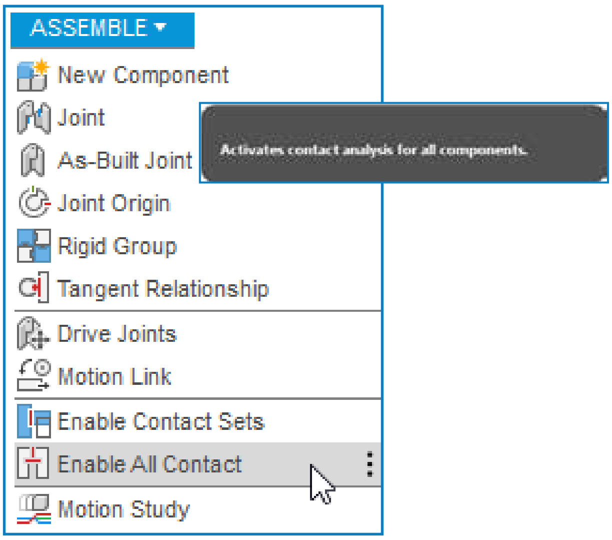
Another advanced method of making connections between components is to use Contact Sets. After activating the command, you can select the components you want to connect. Contact sets allow you to place two or more components in a dependency, simulating a realistic interaction between them.

A major benefit of using contact sets is the ability to optimise the functionality of the components before the manufacturing process. By simulating their interaction, you can check that the components work together smoothly and identify and correct any problems or collisions before real production begins.

It should be noted, however, that activating multiple sets of contacts and animating their movements simultaneously requires a lot of CPU processing power. This can lead to degradation of the animation, as smooth rendering may not be guaranteed. It is therefore recommended that contact sets are only used in animations where a positive connection is particularly important to the production process.

With the targeted use of contact sets, you can efficiently optimise your designs and ensure the functionality of your assembly. By identifying and correcting potential problems early on, you can ensure that your designs meet requirements and function smoothly before they go into production.

3.10 ENABLE ALL CONTACT



By using the „Enable All Contact“ command, you can temporarily deactivate the contact analysis if, for example, you want to perform certain animations where the real interaction of the components is not of crucial importance. This reduces the computing load and allows the animation to run more smoothly.

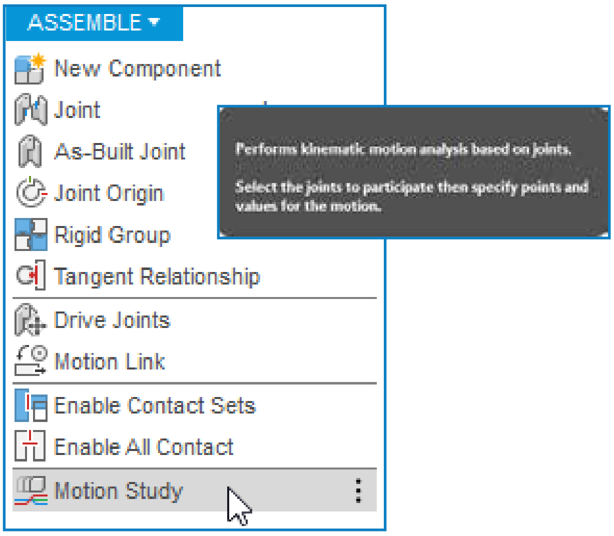
However, it is important to note that contact analysis can still be of great importance in other phases of the design process, such as functionality testing or collision checking. Therefore, you should target the Enable All Contact command and enable or disable it as required to optimise the performance and efficiency of your assembly.

3.11 MOTION STUDY

The "Motion Study" in Fusion 360 is a powerful feature that can be used to create animated motion sequences and simulations of assemblies. It allows users to simulate and visualise the movement and interaction of components in an assembly.

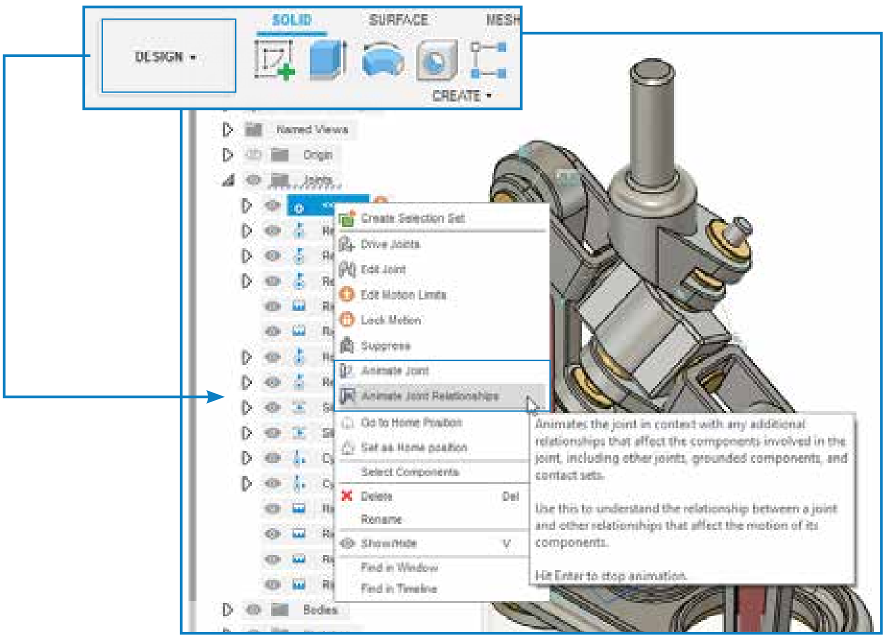
"Motion Study" allows you to define different types of motion, such as rotations, displacements and rotations, and to specify their temporal sequences precisely. This allows you to perform complex kinematic simulations to study the behaviour of mechanisms and machines.

The "Motion Study" is particularly useful for checking the functionality, performance and stability of designs before they go into production. It also helps to identify and resolve potential problems or collisions at an early stage, avoiding costly mistakes in the actual implementation.



3.12 HOW DO I CREATE AN ANIMATION IN THE DESIGN?

To answer the question adequately, it is important to define the term „animation". In this chapter we focus on the animation of movements directly after adding joints to the components. In doing so, we limit the angles and/or lengths of the joints and ensure that the movements are precisely coordinated.



When you add components to a body in the main ‚Design' menu, you must define their position in relation to each other, the type of joint and the direction of movement. The joints created are stored in the structure tree as ‚Joints'. Here you can, for example, create drive joints, suppress joints, lock movements or edit their restrictions. You can also animate individual joints or joint relationships to visualise the behaviour of your constructions. Joint animation is particularly useful for simulating the behaviour of mechanisms and assemblies during assembly and checking their functionality.

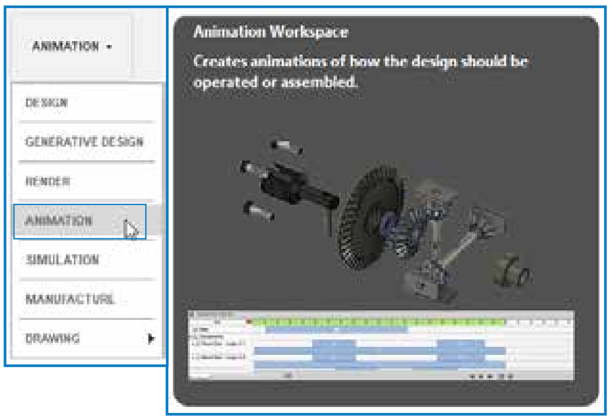
3.13 CREATE AN ANIMATION IN THE MAIN ANIMATION MENU

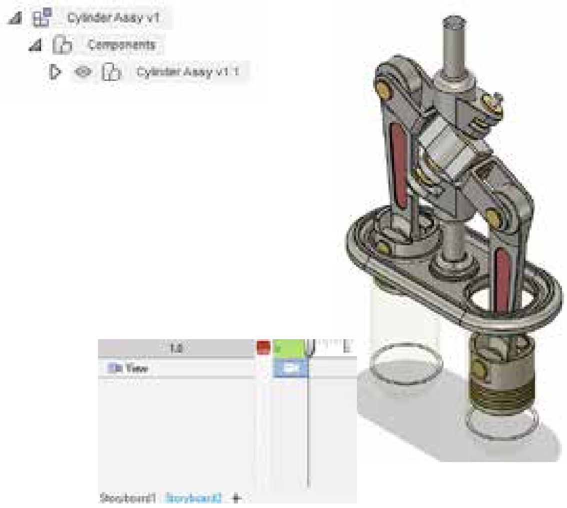
The Animation tab in the main menu allows you to animate the assembly in video quality. From the animated assembly of a module to detailed motion sequences, you can create anything you like. These animations can be used for a variety of purposes, including presentations, product demonstrations and training.

The „Animation" function offers a variety of possibilities to create professional animations. You can set the camera perspectives in desired sequences and speeds to present your construction from different angles and create a dynamic visual experience. In addition, you have the freedom to control the movements of individual components and animate them in synchronised sequences.

This feature allows you to bring your designs to life with realistic movements and simulate complex mechanisms. You can check the functionality of your assembly, identify potential problems and make optimisations.

Whether for customer presentations, internal training or internal design reviews, animation is a versatile tool that allows you to visualise your concepts in a vivid and compelling way.





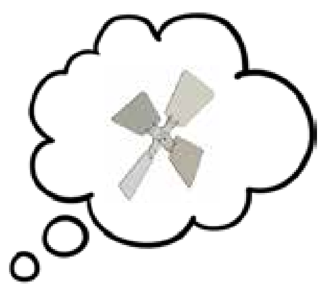
Unit 4

From the idea to the real object: Creation and development with cad and prototyping with a 3D printer

Taking an idea from conception to realisation as a physical object can be an exciting and inspiring journey. In this chapter we will take an in-depth look at the process of how an idea is realised in the world of technology. We will cover the steps from the creation of an idea, through its development using computer-aided design (CAD), to the realisation of a prototype using a 3D printer. This innovative approach allows us to bring creative concepts into the real world and teach students valuable skills for their future careers.

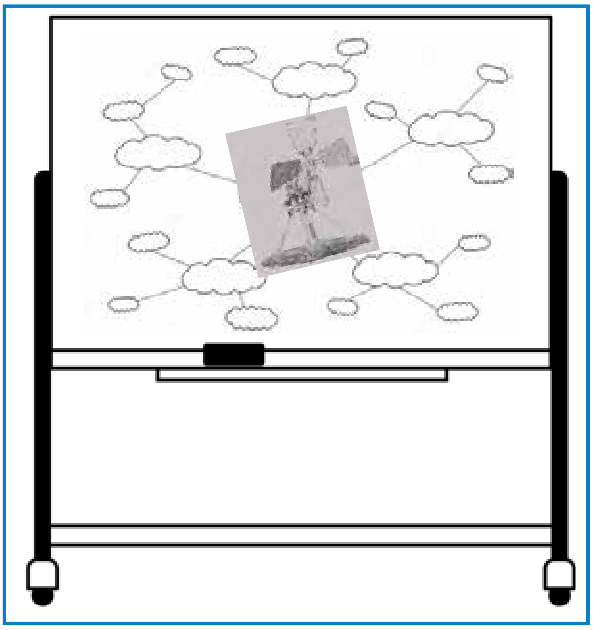
4.1 THE EMERGENCE OF AN IDEA

In the exciting journey from idea to real object, the first and perhaps most important step is the emergence of a creative idea. Ideas can come from a variety of sources and are often the result of individual experiences, personal interests or the need to solve a particular problem. As teachers, we want to encourage our students to unleash their creativity and be open to new ideas and challenges.



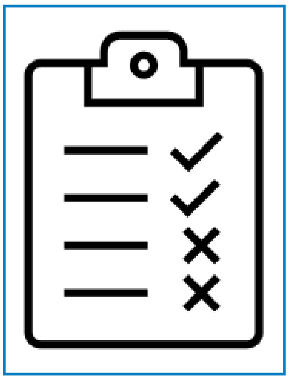
4.2 SPARKING THE FLOW OF IDEAS - BRAINSTORMING AND CONCEPT DEVELOPMENT

In the process of developing ideas, brainstorming is one of the most powerful techniques. It involves collecting ideas, sketching them out and discussing them in groups to further elaborate and refine them. By freely expressing their thoughts and sharing ideas in a group, students can allow their creativity to flourish and create a collaborative environment that leads to the emergence of innovative ideas.

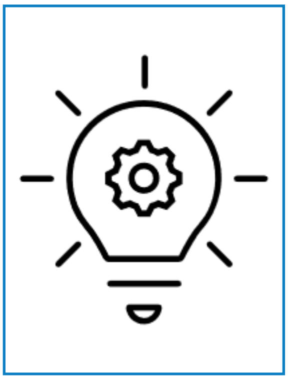


4.3 SELECTION AND EVALUATION OF IDEAS - FROM POTENTIAL TO IMPLEMENTATION

After collecting a variety of ideas, it is important to select the most promising ones. Students are encouraged to evaluate and analyse their ideas in order to select those that have the greatest potential to be turned into reality. Criteria such as feasibility, relevance, degree of innovation and personal enthusiasm can play a role. It is essential that students understand that not every idea can be implemented immediately, but that it can be steered in the desired direction through iterative thinking and further development.

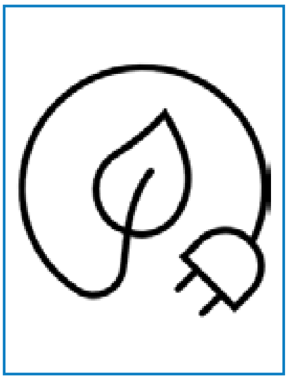


4.4 THE ROLE OF OPENNESS AND INSPIRATION



Openness to new ideas is key to the development of innovative concepts. Students should be encouraged to explore their interests and curiosities and be inspired by their environment. Creative solutions to problems can often be found in everyday situations or in nature. This ‚out-of-the-box' thinking and the ability to be open to new stimuli encourages the emergence of unique ideas.

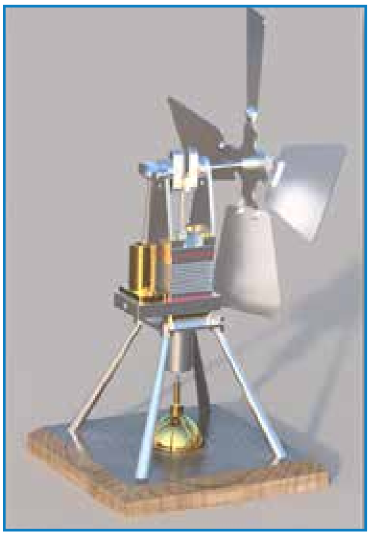
4.5 PRACTICAL EXAMPLES - FROM IDEA TO SUCCESS



Let's imagine that an inventor, looking carefully and curiously at the everyday world, sees a certain need or an unsolved problem. Instead of seeing this obstacle as a mere difficulty, the inventor sees an opportunity to develop an innovative solution. The idea for a ground-breaking product can come from this simple everyday observation.

The simplicity and authenticity of such an idea is often the key to success. It shows us that great innovations do not necessarily depend on complex theories or technical expertise, but are rooted in the observation of life itself. There is an important message in this insight. Each of us can be a true idea generator by being aware of our surroundings and developing creative approaches to solutions.

4.6 PRACTICAL EXAMPLES - IDEA DEVELOPMENT AT SCHOOL



Practical exercises are an ideal method for actively involving the students in the process of developing ideas. These exercises not only promote the creativity of the students, but also enable them to engage with the topic in a lively and committed way.

One way is to have students work in groups or individually on creative projects. By recording their ideas in writing or visually, they can concretise their thoughts and make their visions tangible. This practical experience not only strengthens their creative skills, but also promotes self-confidence and initiative in each individual.

It is important to encourage students to explore and try out different ideas. After all, it is only through experimentation and trial and error that they can expand their imagination and get the flow of ideas going. Mistakes should be seen as part of the learning process, as they often lead to new insights and improved approaches to solutions.

To make the practical exercises even more effective, the students could realise their ideas and projects in the form of prototypes. This is where 3D printing comes in as a valuable tool to turn ideas into reality. Students can create their designs using CAD software and then produce physical models using a 3D printer.

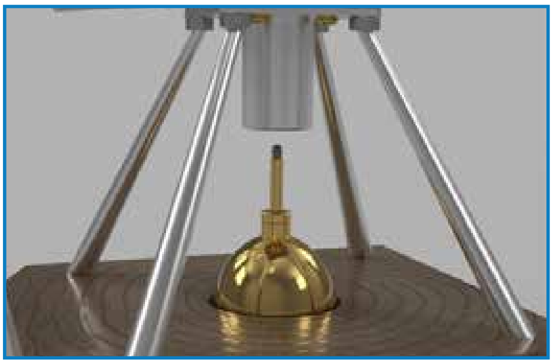
3D printing gives students the opportunity to take their ideas into their own hands and take their creativity to a new level. It is impressive to see real objects being created from digital designs. This not only strengthens technical understanding, but also gives students a sense of self-efficacy and a tangible connection to the real world.

These experiences are not only invaluable for the school environment, but also prepare students for their future professional challenges. The ability to develop ideas and turn them into reality using CAD and 3D printing becomes a valuable tool for their personal and professional development. It enables them to develop innovative solutions and take an active role as creative designers in the technological future.

Unit 5

From fire to power – CAD implementation of a sterling enging with an open flame as heat source

The Sterling engine is a fascinating example of an alternative drive method that can be used in model making. Its mode of operation is based on the principle of the thermodynamic cycle and uses an external heat source to generate mechanical work.



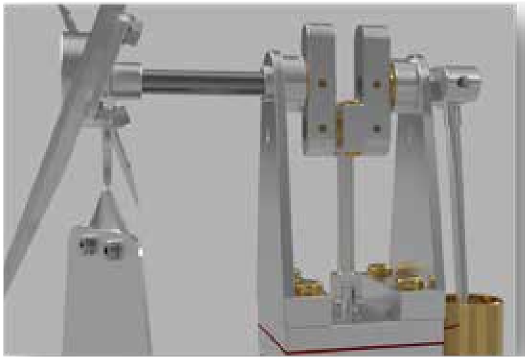
The design of a Sterling engine is comparatively simple, and its operation does not require any additional coolant. It consists of two cylinders connected by a heat exchanger (regenerator). Inside the cylinders are pistons that move synchronously and in opposite directions. Flywheels are attached to the pistons, which help to stabilise the rotation.

The external heat source, in this case an open flame, is placed under the heat exchanger. Due to the effect of the heat on the regenerator, the air in cylinder A heats up and expands, causing the piston to move upwards. At the same time, the air in cylinder B cools down, causing the piston in this cylinder to move downwards. This creates a pressure difference between the cylinders.

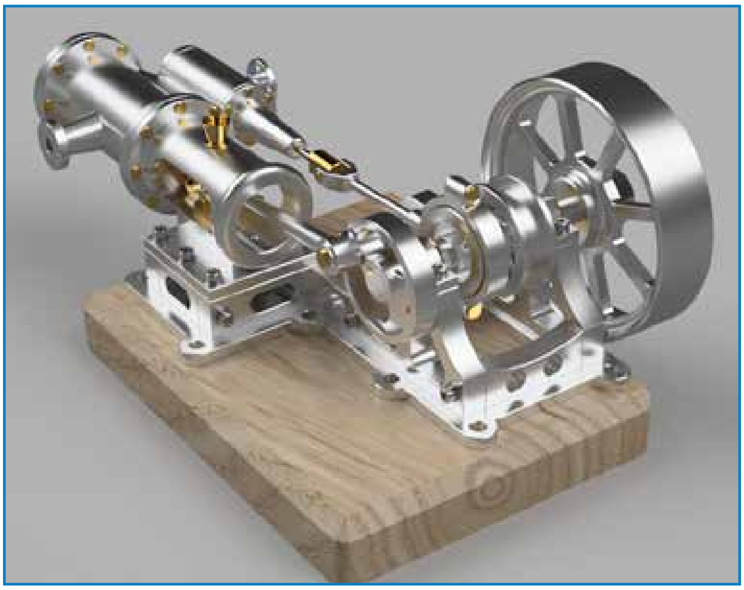
Once the piston has reached its top position, a sliding mechanism is activated that moves the piston into the other cylinder. Now the process starts again, moving the piston upwards in the other cylinder and repeating the cycle.

The key to the efficiency of the Sterling engine is the regenerator, which stores heat energy during the process and transfers it between the cylinders. This improves efficiency by reusing some of the heat energy that would otherwise be lost.

The motion of the piston in a Sterling engine can easily be transferred to a flywheel as a rotary motion.



This flywheel can now perform mechanical work, for example drive a shaft or operate a generator to produce electrical energy.

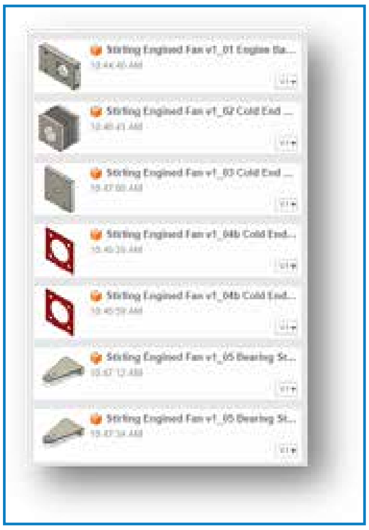


The Sterling Model Engine provides an educational opportunity to understand the basics of thermodynamics and heat exchange. Students can experience first-hand how this fascinating engine works and explore its potential applications in the field of alternative energy sources.

In addition, the use of a Sterling engine in model making provides important engineering and mechanical skills. Students can experiment to increase the efficiency of the engine and test different configurations. This hands-on learning develops their problem-solving skills and creativity in optimising the engine‘s performance.

5.1 STIRLING ENGINED FAN

In this chapter we will combine existing components into a Stirling engine and define their movements as joints. In order not to extend the scope unnecessarily, the steps will be divided into several sequences, so that not all components are presented individually.

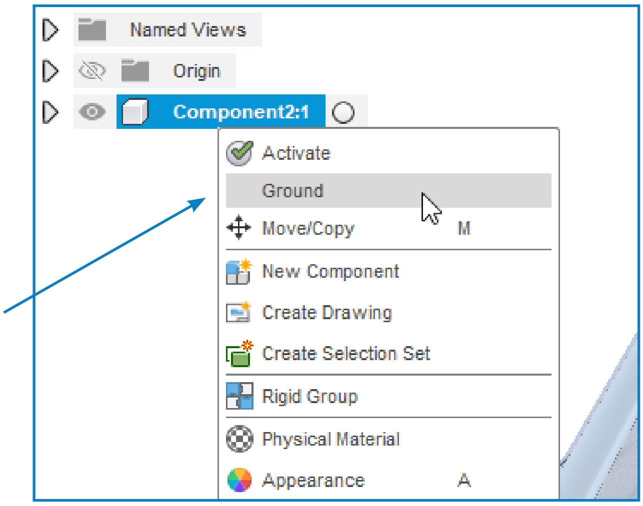


When you start creating the assembly, it is advisable to use components that have already been constructed. Think about the order in which you want to create the assembly, which boundaries you need to set and between which movements a link should be created.

Let us now start with the base plate by opening the body with a double click of the left mouse button.

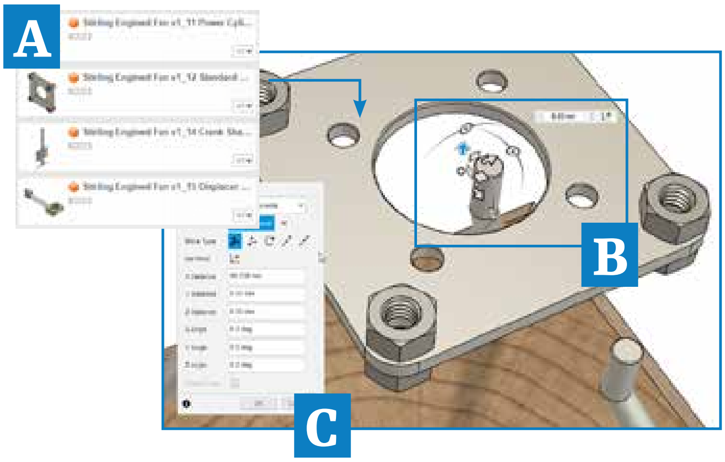
5.2 FIXING THE ASSEMBLY

Now convert the basic body into a component and fix it with the command „Ground" to avoid unintentional shifting of the components against each other. The fixation can be removed at any time if you want to make further adjustments. This gives you full control over the movements and positions of the components and allows you to design your assembly precisely

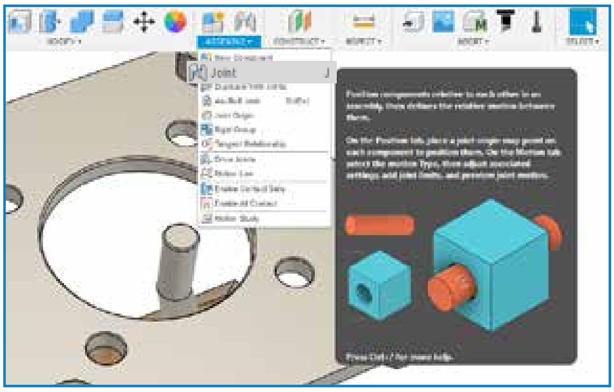


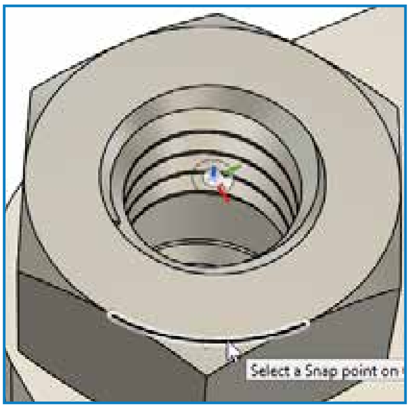
5.3 INSERT JOINTS

To add the following component to the assembly (see Figure A), simply use drag and drop. Position this component close to the component to be connected, either by using the X, Y and Z coordinates (see Figure B) or by entering the coordinates in the input field provided (see Figure C). Confirm by either clicking on ‚OK' or pressing the ‚Return' key

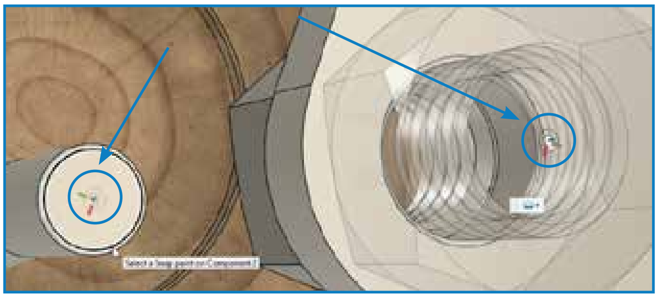


To create the joint between the two components, select the „Joint" command in the „Assemble" menu. With this command you now have the possibility to create joints between the two components and link their movement. This allows you to realistically simulate and optimise the functioning and interaction of the components in your assembly. The „Joint" command allows you to position the components exactly according to your requirements and to control their movement precisely.



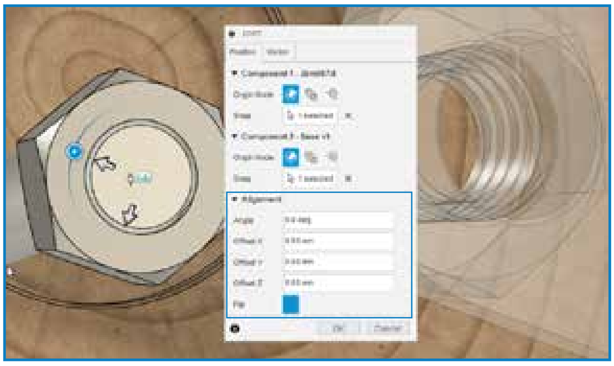


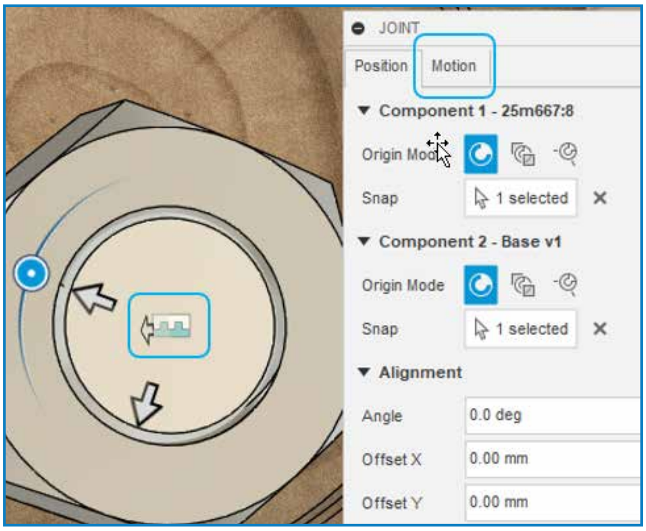
To add a component to an existing assembly, you must first select the component to be added. The movement is then in the direction of the existing component or assembly. In this example, as two cylinder shapes are to be joined, select the centre of the component to be moved to ensure precise alignment.



To define the position of the component to be added, we need a target plane on the existing component as a reference. This target plane serves as a starting point for the precise placement of the new component. By selecting the target layer, we can ensure that the component to be added is placed in the desired position and orientation..

After the respective levels have been determined, they are moved on top of each other to move the component into the desired position. You have the option to precisely determine the position changes manually via the input field. This precise setting allows you to align the components exactly and achieve the desired arrangement in your assembly.

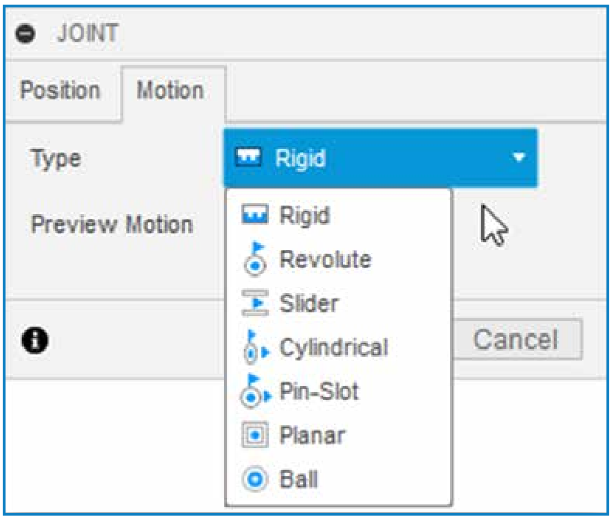




In the input field there is the subitem „Motion", which allows you to change the joint connection. By selecting this subitem, you can specifically adjust and optimise the movements and interactions of the components in your assembly. This opens up the possibility for you to simulate different motion sequences.

The functions and applications of the various joint types in CAD, such as Rigid, Revolute, Slider, Cylindrical, Pin-Slot, Planar and Ball, are fundamental to defining and controlling movement between components in an assembly.

These joints make it possible to model and simulate the interactions and connections between the individual components in a virtual environment, just as in a mechanical system. Below I explain the function and application of each type of joint:



1. Rigid Joint: Function: The „Rigid Joint“ provides a fixed and immovable connection between two components. It restricts any movement or rotation between the components.

USE: „Rigid Joints“ are often used to hold components in a fixed position when no movement between them is required.

2. Revolute Joint: Function: The „Revolute Joint“ allows rotation around a common axis between two components. It restricts other movements along this axis.

USE: „Revolute Joints“ are ideal for rotary movements of parts, such as opening and closing a door or turning a crank.

3. Slider Joint: Function: The „Slider Joint“ allows linear movement along a common axis between two components. Other movements vertical to the axis are restricted.

USE: „Slider Joints“ are suitable for linear movements, such as pushing a drawer or extending and retracting a telescopic tube.

4. Cylindrical Joint: Function: The „Cylindrical Joint“ combines the functions of the „Revolute Joint“ and „Slider Joint“. It allows for both rotation and linear movement along a common axis between the components.

USE: „Cylindrical Joints“ are suitable for mechanisms that require both rotation and linear motion, for example in hydraulic cylinders.

5. Pin-Slot Joint: Function: The „Pin-Slot Joint“ enables a component to be moved along a given rail or slot.

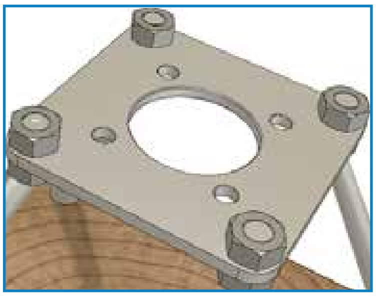
USE: „Pin-Slot Joints“ are often used in sliding mechanisms to guide the movement along a specific path.

6. Planar Joint: Function: The „Planar Joint“ allows movement along a fixed plane between the components. Other movements vertical to the plane are restricted.

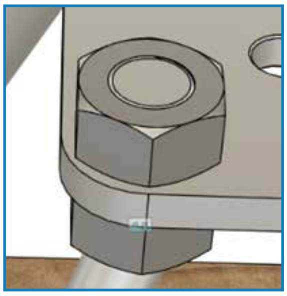
USE: „Planar Joints“ are used in mechanisms where movements are required within a plane, such as hinges.

7. Ball Joint: Function: The „Ball Joint“ allows free rotation in all directions around a common centre. Application: „Ball Joints“ are ideal for mechanisms that require a wide freedom of movement in all directions, such as in the control of robot grippers.

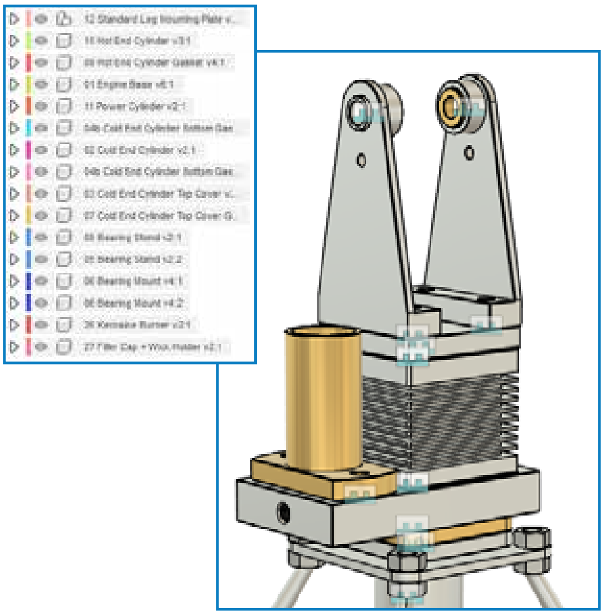
USE: „Ball Joints“ are ideal for mechanisms that require a wide freedom of movement in all directions, such as in the control of robot grippers.



The „Standard Leg Mounting Plate“ is now firmly attached to the base, so that their position relative to each other cannot be changed.

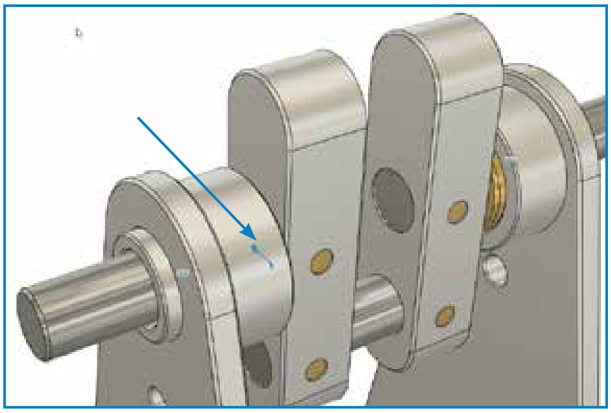


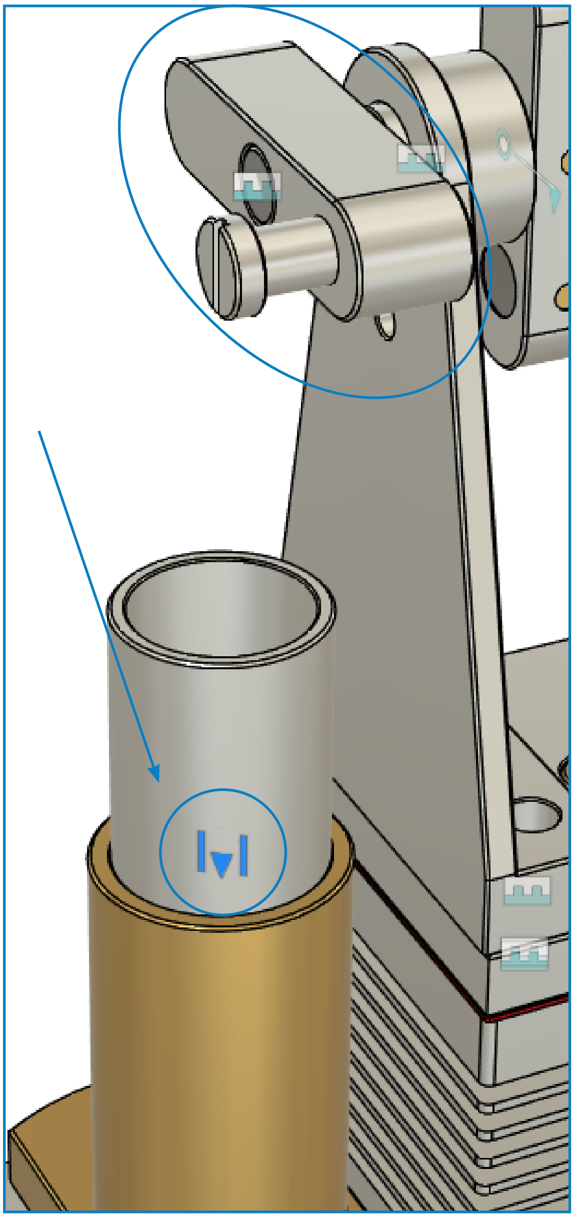
All the components shown have been combined with rigid joints. These rigid joints form the basic structure of the assembly and enable reliable fixation of the individual parts. Later in the project, additional elements such as screws or pins can be added, which are also rigidly combined to further stabilise the assembly and allow for additional functionality. This step-by-step addition and expansion of the construction gives you the flexibility to adapt and optimise the design according to requirements.



5.4 INSERTING MOVABLE COMPONENTS

Insert the already prefabricated component „Stirling Engine Fan" into the assembly as described before. Then create a joint connection between this component and the existing assembly. For this, use the „Revolute" joint type, which enables a rotational movement around a fixed axis. With this joint you can define the movement of the Stirling Engine Fan in the assembly and thus its interaction with the other components.



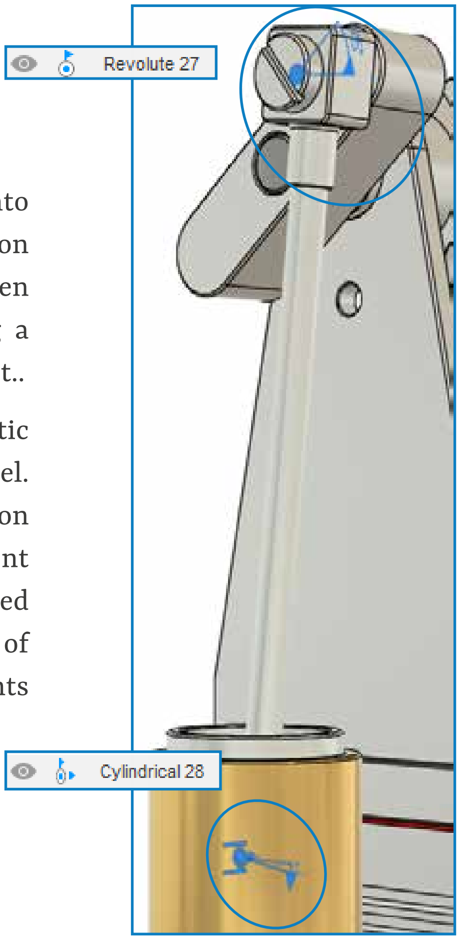


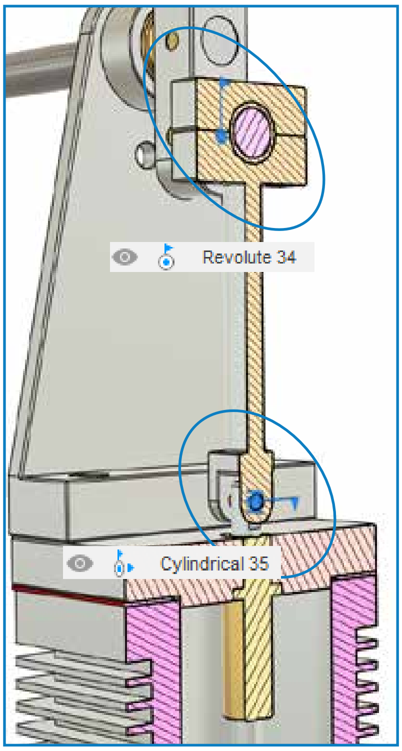
After the component „Power Piston Crank + Pin" has been firmly inserted into the assembly, we will now connect the „Power Piston" component to the „Power Cylinder" and create the possibility to move it along a slider axis. For this we will use the command „Slider Joint". When creating this joint, you must again select two planes that can be shifted against each other. This displacement can be limited in both positive and negative directions.

By cleverly using the ‚Slider Joint' you can simulate the movement of the piston in your Stirling engine and realistically replicate its functionality. This joint allows you to optimise the mechanics of the engine and test different configurations to ensure efficient and smooth interaction of the components. This allows you to analyse the performance of your Stirling engine in a virtual environment and make improvements where necessary.

In the following step, we insert the „Power Piston Conn Rod“ into the assembly and create both a connection with the „Power Piston Crank + Pin“ using a „Revolute“ joint and a connection between the „Power Piston“ and the „Power Piston Conn Rod“ using a „Cylindrical“ joint. It does not matter which joint is created first..

Through these clever connections, we create a realistic kinematic representation of the components in our Stirling engine model. The „Revolute“ joint allows the rotational movement of the piston crank, while the „Cylindrical“ joint allows the linear displacement of the Power Piston along the piston rod. These carefully designed joints allow us to accurately simulate the mechanical operation of the engine and precisely control the movement of the components in the assembly.



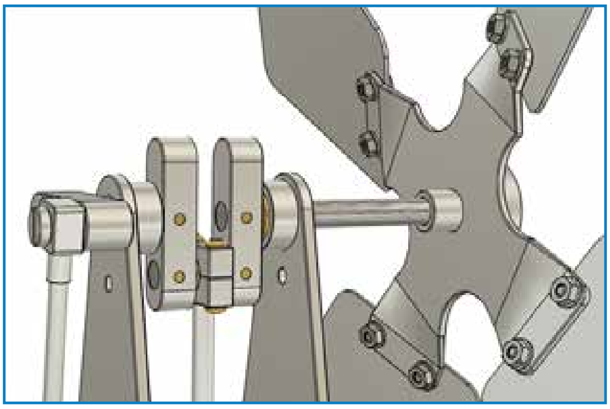


In the next step we connect the „Crank Shaft“ component to the „Revolute Joint“ and the „Displacer Piston Top Cap Rod“ to the „Cylindrical Joint“, similar to the previous step.

Through the clever application of the „Revolute“ and „Cylindrical“ joints, we enable the smooth interaction of the various components in our Stirling engine model. The „Revolute“ joint allows the rotational movement of the crankshaft component, while the „Cylindrical“ joint allows the linear displacement of the Displacer Piston Top Cap Rod along the Displacer Con Rod.

These precise joints realistically reproduce the mechanics of the Stirling engine in our assembly group.

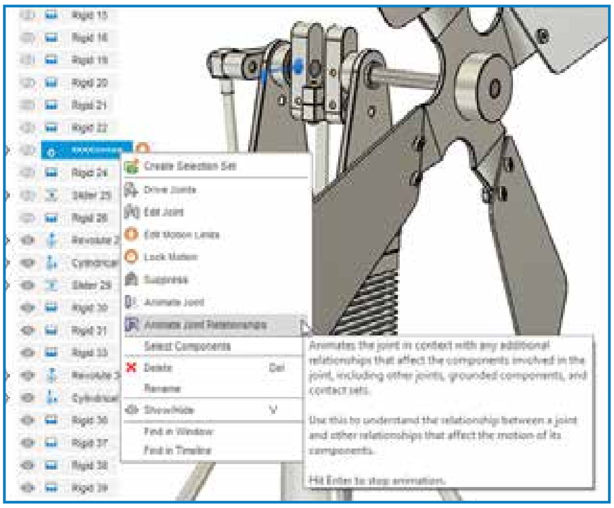
To complete the assembly of our Stirling engine, we now rigidly connect all the remaining components, screws and pins to the assembly. This ensures that all parts are fixed in the desired position and cannot move against each other unintentionally.



The careful use of rigid connections is of great importance to ensure the overall functionality and stability of our assembly. By fixing the individual parts, they are brought together to form an integrated and functional system that enables the movement of the various components in our Stirling engine.

It is important to provide all components with the correct rigid connections to ensure a realistic simulation of the engine‘s operation.

At the end of our project, we will create an animation to ensure that our assembly performs the required movements smoothly, with no collisions or misalignment of components. The animation will allow us to realistically simulate the functionality and movement of our Stirling engine and identify any errors or inconsistencies early on.



To simulate individual movements, we can use the ‚Animate Joint‘ command. This allows us to move the assembly step by step into different positions and check the movement of each joint. This way we can make sure that the joints are working properly and performing the desired movements.

In addition, we have the option to set the entire assembly in motion and simulate the interaction of all the joints present. To do this, we use the “Animate Joint Relationships” command. This allows us to animate the complex motion of the assembly involving all the existing joints. This gives us a comprehensive understanding of how our Stirling engine works and allows us to identify any problems or limitations. Animation is a valuable way to validate and optimize our design before we physically implement it. It helps us identify and fix any potential errors or weaknesses early on, saving time and resources in the later implementation phase.

Unit 6

3D Printer and slicing

In the following chapters, the topic of 3D printing, as well as the appropriate program for printing will now be explained in more detail.

What is 3D printing? This printing process is an additive process where material is added to an object or built from scratch. This is the opposite process to the exciting process of a CNC machine.

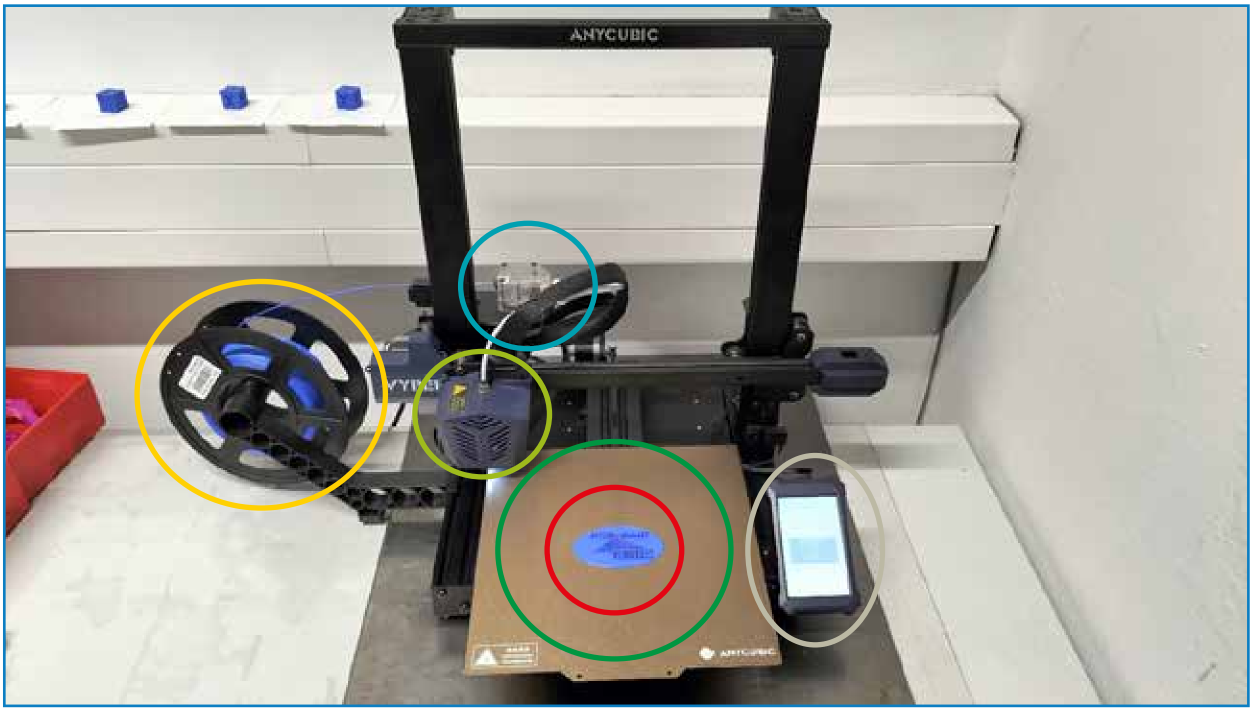
3D printing mostly uses plastics, or more specifically, thermoplastics. These are melted and repositioned on a print bed, and then cooled. The plastic is fused together layer by layer until a finished model is created at the end.

For the printer to know exactly how to proceed, a 3D model, which has been saved as an STL file, must first be prepared in a slicer.

As an introduction to the topic, the program „Ultimaker Cura 5.2.1“ is used.

The Cura slicing program ensures that the desired model is provided with the correct parameters. These include the temperature, the speed, and the position of the workpiece. When the settings have been adjusted as desired, the program can convert the model into a GCode file and prepare it for the 3D printer. After conversion, the file must be saved to a disk which needs to be connected to the printer. The 3D printer will then do the rest of the job.

6.1 CONSTRUCTION OF THE 3D PRINTER



I. Filament, the material that is melted and formed into the new shape.

II. Extruder: Here the filament is clamped and is automatically transported to the print head by means of hoses.

III. Hot end: This melts the filament and emits it via a 0.4mm nozzle. The nozzle can be replaced with other sizes.

IV. Printing bed: This is the base plate on which the melted filament is placed. This is usually also heated.

V. Finishing model: The model that was previously sliced is now printed onto the base plate.

VI. Control panel: With the help of the control panel, separate settings can be made on the printer, such as the filament change, preheating or leveling.

6.2 THE SOFTWARE CURA: A POWERFUL TOOL FOR 3D PRINTING

Cura is a popular open-source software specifically designed for 3D printing. The program serves as a slicer, which means it converts 3D model files into instructions that the 3D printer understands and uses to create the physical object. Here is an overview of the main features and functions of Cura:

1. 3D Model Preparation: Cura allows users to import 3D model files in various common formats such as STL, OBJ, and 3MF. These files contain the 3D geometry of the desired object.

2. Print Settings: Cura provides a wide range of print settings that can be customized by the user. These settings include layer height, print speed, infill density, wall thickness, print temperature, bed temperature, and more. These settings enable users to control the quality and speed of the printing process.

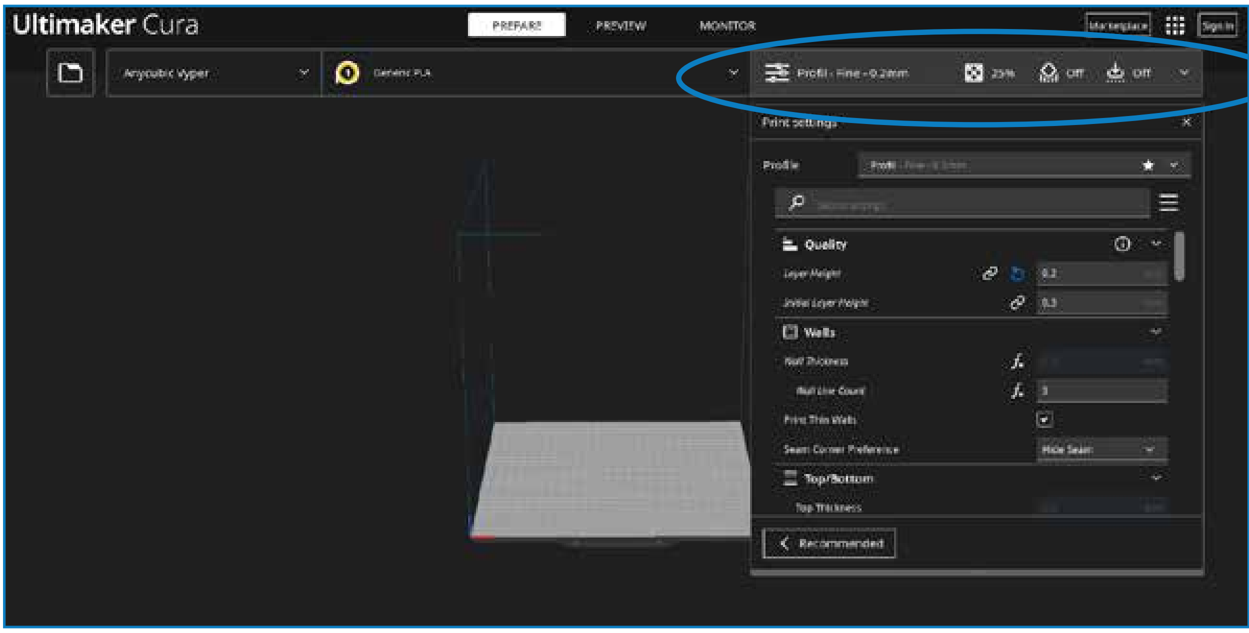
3. Slicing: After the print settings have been defined, Cura uses the slicing algorithm to divide the 3D model into individual layers. These layers are then converted into a G-code file, which the 3D printer can use to print the object layer by layer.

4. Preview and Simulation: Cura offers a preview of the printed object, including a layer-by-layer view. This allows the user to review the design and identify potential errors or issues in the printing process before starting the actual print. There is also a collision detection feature that indicates possible collisions between objects and the print bed.

5. Print Bed Adjustment: Cura allows users to calibrate the print bed and precisely adjust the starting position of the print head. This ensures optimal adhesion of the print to the print bed and prevents potential errors during the printing process.

6. Print Profile Management: Cura enables users to create and save various print profiles for later use. This is useful when different print settings are required for different materials or printing tasks.

6.3 GENERAL

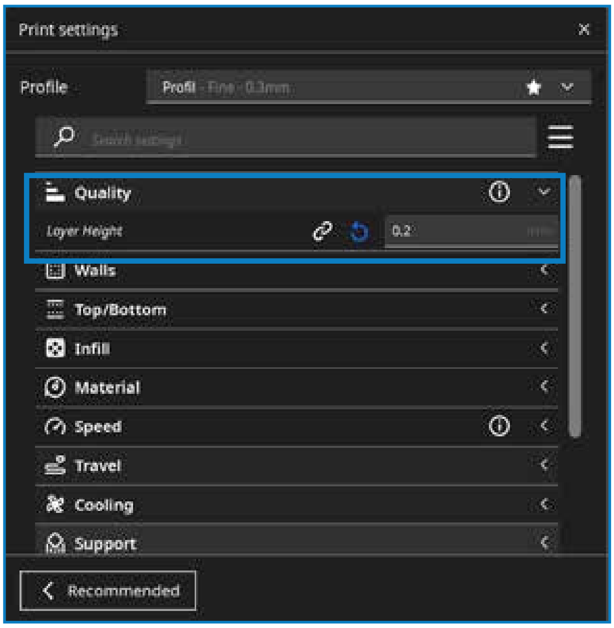


The following settings of the model can be customized:

- Size
- Position on the print bed
- Thickness of the print layers
- Structure of the interior
- Temperature of the print/heating plate
- Speed of printing
- Cooling of the material
- Support structures for printing
- Adhesion of build plate

STL files are required for the slicer!

6.4 QUALITY



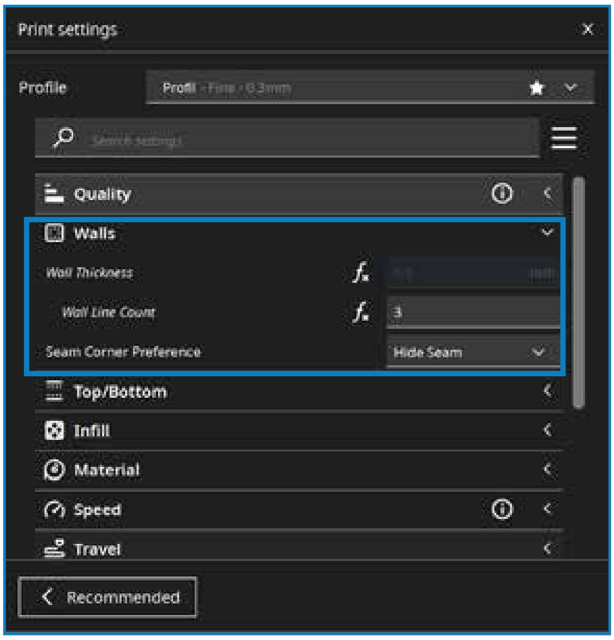
In the 3D printing setup of Cura, you have the ability to adjust the wall thickness and outer contours of your models according to your preferences. Fine-tuning these parameters can enhance print quality and stability. The wall thickness influences the robustness and durability of the printed object, while adjusting the outer contours enables more precise detailing and a smoother finish.

Modifying the wall thickness can also help conserve material and reduce printing time without compromising the structural integrity. Additionally, having control over the outer contours allows you to create precise details and eliminate potential irregularities or gaps in the print.

It is advisable to experiment with various settings and print prototypes to determine the optimal wall thicknesses and contour adjustments for your specific project.

Values for a 0.4 nozzle: 0.10 - 0.30 mm

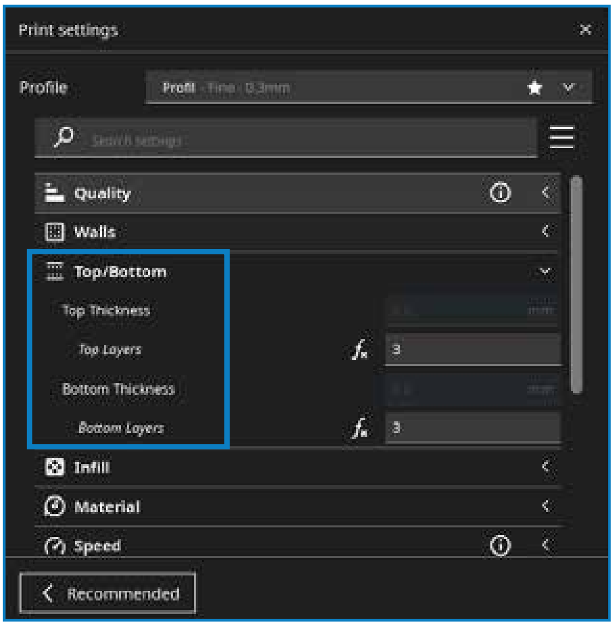
6.5 WALLS



Here, you have the option to adjust both the wall thickness and the outer contours. The wall thickness can be specified either in millimeters or by the number of lines. If you provide the number of wall lines, it will be multiplied by the nozzle diameter. Example: 4 wall lines x 0.4 mm nozzle diameter = 1.6 mm wall thickness. It is important to note that thicker walls enhance print stability, but they also increase the required printing time. The default value for a 0.4 mm nozzle is 1.2 mm or 3 wall lines.

Additionally, you can customize the Z-seam alignment under the walls. The Z-seam represents the point where the nozzle repeatedly starts at the outermost layer.

6.6 TOP/BOTTOM



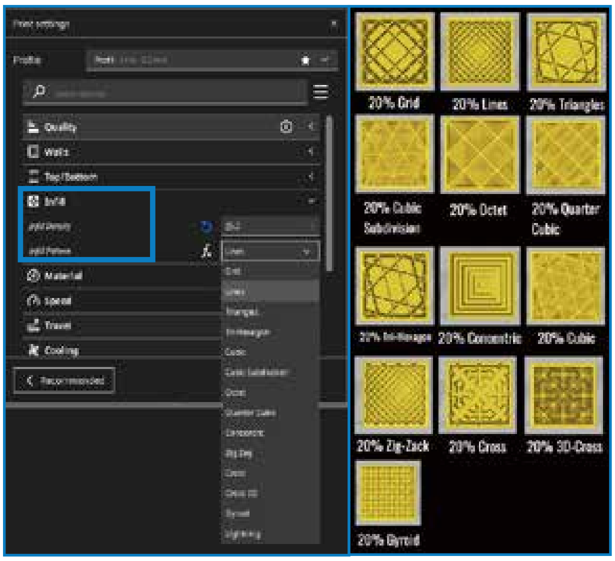
In Cura, you have the option to set the layer thickness in both millimeters and by the number of layer lines. When you specify the number of layers, it will be multiplied by the layer thickness.

More layers generally result in higher print quality and stability as the printed object is composed of more layer slices. However, this also leads to longer printing times since more layers need to be created sequentially.

The default value for the top and bottom layers with a 0.4 mm nozzle is 0.9 mm or 3 layers. This value serves as a good starting point and provides a balance between print quality and printing time. Depending on your project's requirements, you can customize these settings to achieve the optimal outcome.

Fine-tuning the layer thickness allows you to achieve the desired surface finish and level of detail in the printed model. Experiment with different settings to find the best trade-off between printing time and print quality, enabling you to achieve impressive results

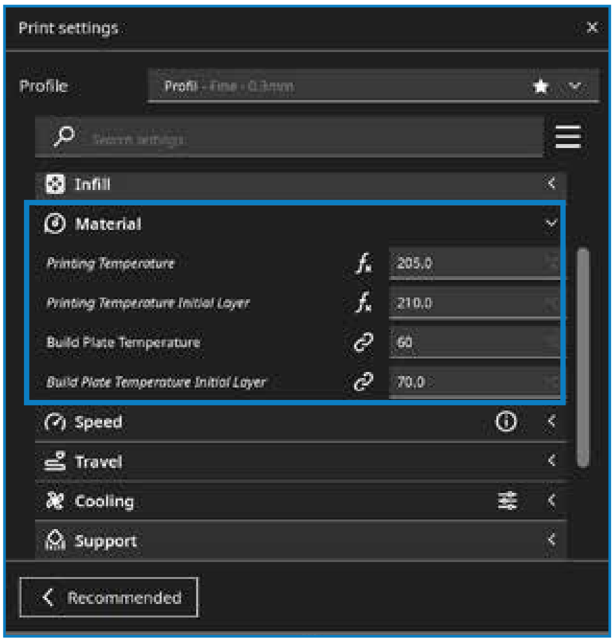
6.7 INFILL



In the „Infill“ category, the infill density of the print is determined, meaning the ratio of material to air in the final product. This infill density is selected as a percentage. Additionally, the infill pattern can also be chosen.

The standard options available for selection are:
Grid, Triangles, Tri-Hexagon

6.8 MATERIAL



The temperature at which the filament is melted is important for printing.

This depends on the filament type.

The temperatures can vary between the manufacturing companies and must be adjusted.

The print bed must also be heated correctly to match the filament to create a high level of strength when printing between the printing plate and model.

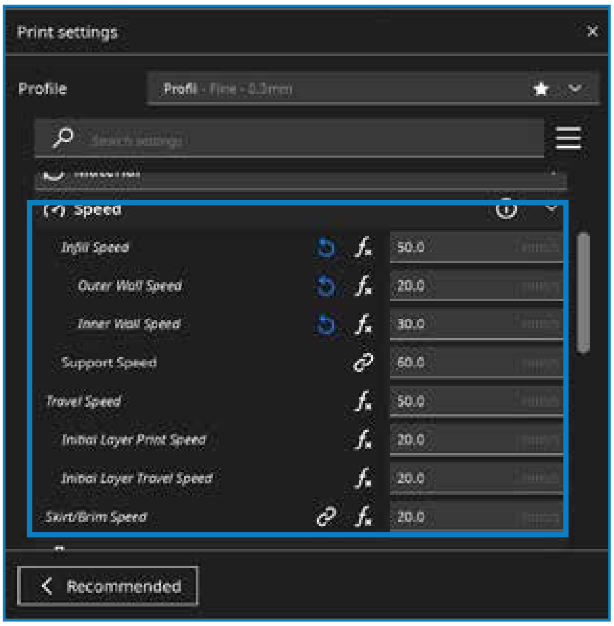
What is 3D printing material PLA made of? PLA (Polylactic Acid) is a popular bioplastic material used for 3D printing. It is composed of natural resources, typically sourced from plant-based origins such as corn starch, sugarcane, or other sugar-rich plants. These renewable resources undergo fermentation to convert them into lactic acid, which is then used to produce Polylactic Acid (PLA).

PLA is biodegradable and environmentally friendly, making it one of the most commonly used materials in 3D printing, especially in applications that require an eco-friendly alternative to traditional plastics.

Printing temperature:		**Printing bed:**	
	- PLA: 190°C – 220°C		60°C – 80°C
	- PLA+: 200°C – 225°C		60°C – 80°C
	- PETG: 210°C – 250°C		80°C – 100°C
	- TPU: 220°C – 240°C		60°C – 80°C
	- ABS: 210°C – 260°C		100°C – 115°C

For better adhesion to the print bed, the temperature for the first layer can also be increased slightly **(approx. +5-10°C)**. This applies to the printing temperature as well as to the print bed.

6.9 SPEED



Under these settings, all speeds required for printing can be adjusted. These are divided into infill speed, wall speed and support structure speed. Basically, the slower you print, the fewer printing errors will occur.

The infill speed sets the speed for the inner filling of the model.

The wall speed for the inner wall and the outer wall should be adjusted separately. This is because the outer wall should be printed about 10-20mm/sec slower than the inner wall to create a better surface.

The support speed can be similar to the filling speed.

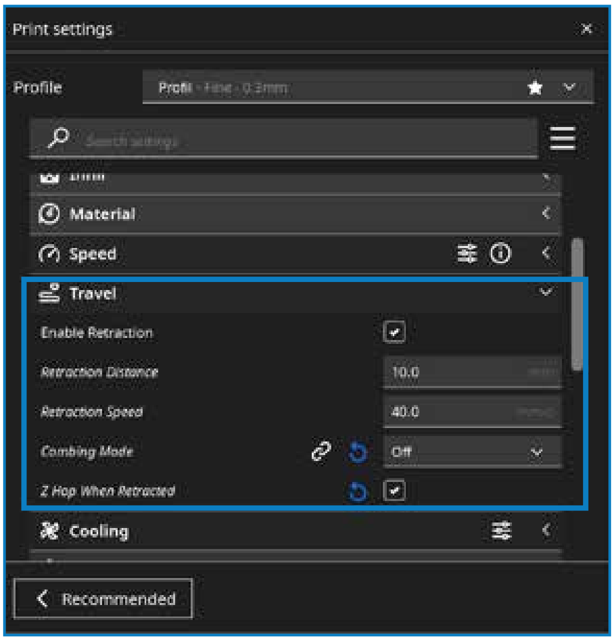
Again, the speeds should be reduced for the first layer to create more strength at the print bed.

Skirt/Brim speed should also be kept low. For more information on Skirt/Brim, see number 6.13

	filling speed	**inner wall speed**	**outer wall speed**	**support speed**	**initial layer speed**
PLA	70 mm/Sek	50 mm/Sek	40 mm/Sek	60 mm/Sek	5-15mm/Sek
PLA+	70 mm/Sek	50 mm/Sek	40 mm/Sek	70 mm/Sek	5-15mm/Sek
PETG	50 mm/Sek	30 mm/Sek	25 mm/Sek	30 mm/Sek	5-15mm/Sek
TPU	30-40 mm/Sek	30 mm/Sek	25 mm/Sek	30 mm/Sek	5-15mm/Sek
ABS	60 mm/Sek	50 mm/Sek	40 mm/Sek	50 mm/Sek	5-15mm/Sek

The speed may vary by manufacturer.

6.10 TRAVEL



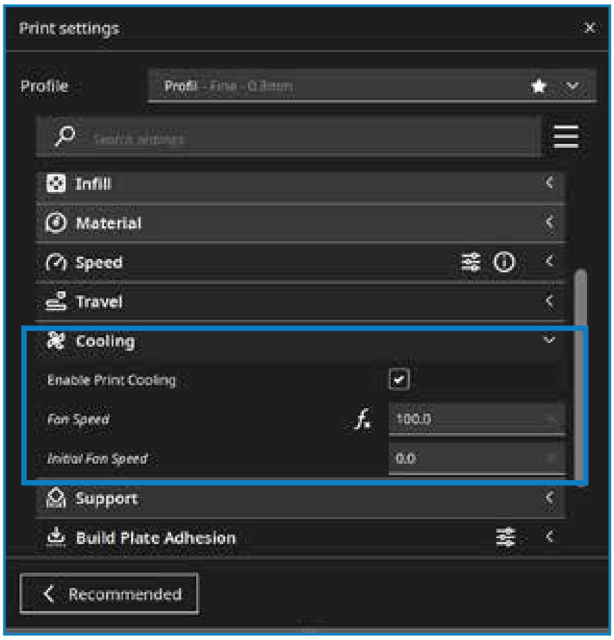
In the **Movement tab**, all movements that are responsible for printing are set. An example is the movement of the filament from the motor to the melting nozzle. This is also defined as the **feed**.

The feed should be activated to prevent filament formation during printing. Here, a piece of filament is pulled back when the print head wants to take a new position. The speed can also be adjusted for this purpose.

Default value: 40mm/sec

Combing mode should always be switched off. For a **better surface** on the workpiece, the **Z-jump** can also be switched on during feeding. This has the effect that when a feed occurs, the print head moves 1mm higher and thereby prevents the model from being damaged during the next movement.

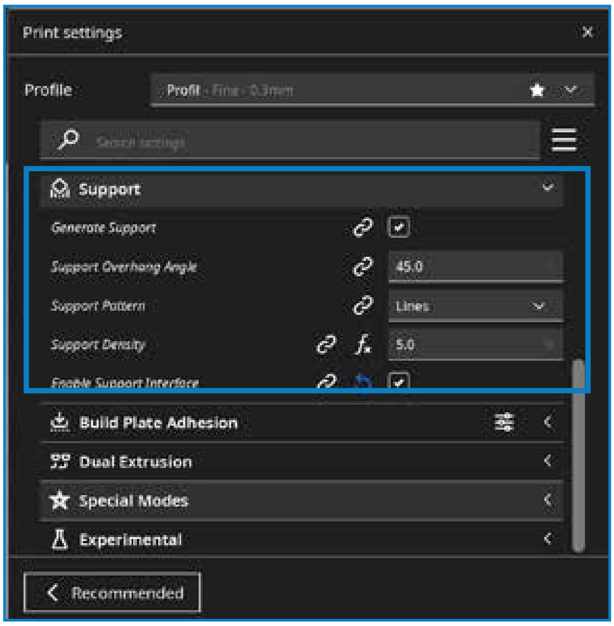
6.11 COOLING



After the printer has placed the filament on the print bed, it is important that the material quickly drops below the melting temperature. The built-in cooling of the printer is used for this.

The cooling should be set to 0% for the first layer to ensure better adhesion to the print bed.

6.12 SUPPORT



In case the model has overhangs, holes or a complex structure, the support structure function should be switched on. In this case, supporting pillars are printed under the hanging print points. This is because the printer is not able to print structures in the air. The function can also help prevent the project from falling over during printing. After printing, the structure can be removed from the model effortlessly and without damage.Here, the set angle should always **be at least 45° or smaller**. As with the fill structure in 3.5, the shape and density of the structure can be adjusted.

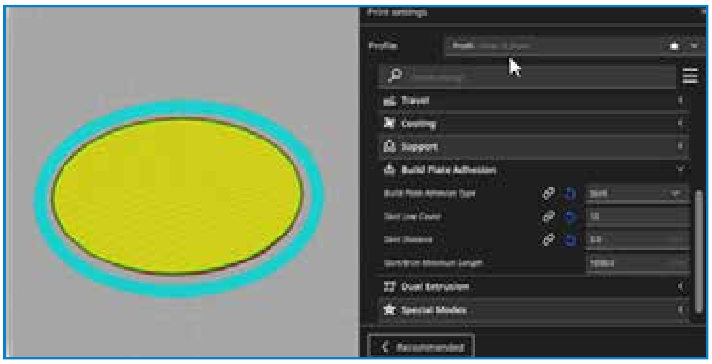
Standard: Mesh with 10% density

To guarantee a better surface on the model, the support structure interface should be activated.

6.13 BUILD PLATE ADHESION

Printing plate adhesion is another way of fixing the model more firmly to the printing plate. A distinction is made between 3 types.

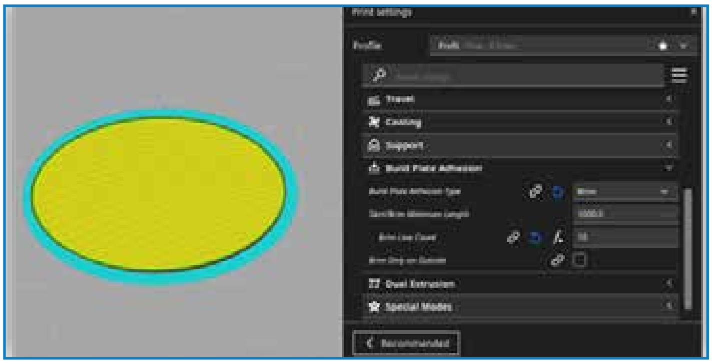
I. Skirt



The skirt is a simple line path around the model and is one layer high. This has the use of checking whether the print bed has been properly aligned. The number of lines printed can be set, as well as the distance to the main body.

Default value: 5 lines with 5mm

II. Brim



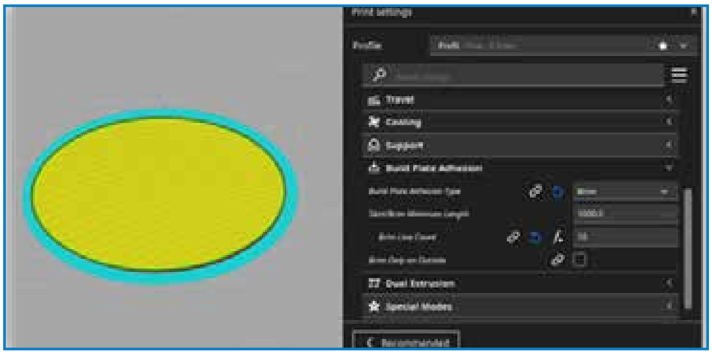
The Brim is like the Skirt, except that it is directly connected to the workpiece. The benefit of this is that the printing error warping can be minimized. In addition, plate strength can be increased on thin workpieces because a larger print area is created. However, the brim must be removed by hand after printing and may cause damage to the printed piece.

Here, too, the number of lines or the line width can be adjusted.

In addition, the brim can also be generated only on the outer sides of the print, if necessary.

Default value: 15 lines or 6mm

III. Raft



Raft is a special printing plate adhesion method in which a small „island“ is first printed under the actual model. This has the benefit of creating a uniform plane, which can prevent warping, for example, and easy release from the printing plate after it has cooled down.

The raft consists of 3 layers. Its printing speed corresponds to the speed of the first layer. However, this can also be set individually.

The advantage of this is a constant and fixed pressure, although it has a significantly increased printing time. Good use when several smaller models are printed, where the brim function cannot find use.

Unit 7

Printer preperation

Before a CAD model can be printed, the following must be prepared at the printer and be checked regularly:

Levelling the print bed

The most important point for clean adhesion of the filament to the print bed is levelling. Here, the print head must be perpendicular to every point of the print bed and have a certain distance.

Instruction:

There are several ways of levelling.

1. Autolevelling: in this case, the autolevelling option is selected in the settings of the device. The printer will then level the bed automatically. Not every printer has this function.
2. Manual levelling: For this, the printer must be switched on and the Home Z-axis option must be selected in the settings. Here the print head moves down to the zero point. A sheet of paper should be pushed between the print plate and the print tip. Now check whether the sheet can be moved freely at the corners and in the middle of the print bed or not. If the sheet can be moved, the printer must be readjusted using the adjustment wheels on the print bed. If the sheet does not move, the plate must be loosened. At the end of levelling, the sheet must drag between the print bed and the head.

When levelling has been performed successfully, it does not have to be performed again unless the printer/print bed has been moved a lot or has been transported.

Cleaning the print bed

To guarantee good printing strength, the printing plate must be cleaned before each print. In the best case, with isopropanol or other high-proof alcohol. Other cleaning agents can also be used, as long as the print bed is free of grease.

If necessary, a printing plate mat can also be used. These are usually equipped with a special surface that eliminates the need for cleaning.

Firm stand of the printer

It is also important to ensure that the printer is placed on a solid surface. Vibrations from the outside and from the printer itself, due to an unsolid stand, can degrade the print quality.

Air conditioned room

To avoid distortions during printing, the printer should be located in a room where there are hardly any drafts, and it is not too cold. At best, a printer with its own printing chamber should be used or it should be used in a cabinet.

Filament choice

The filament type plays an important role in printing.

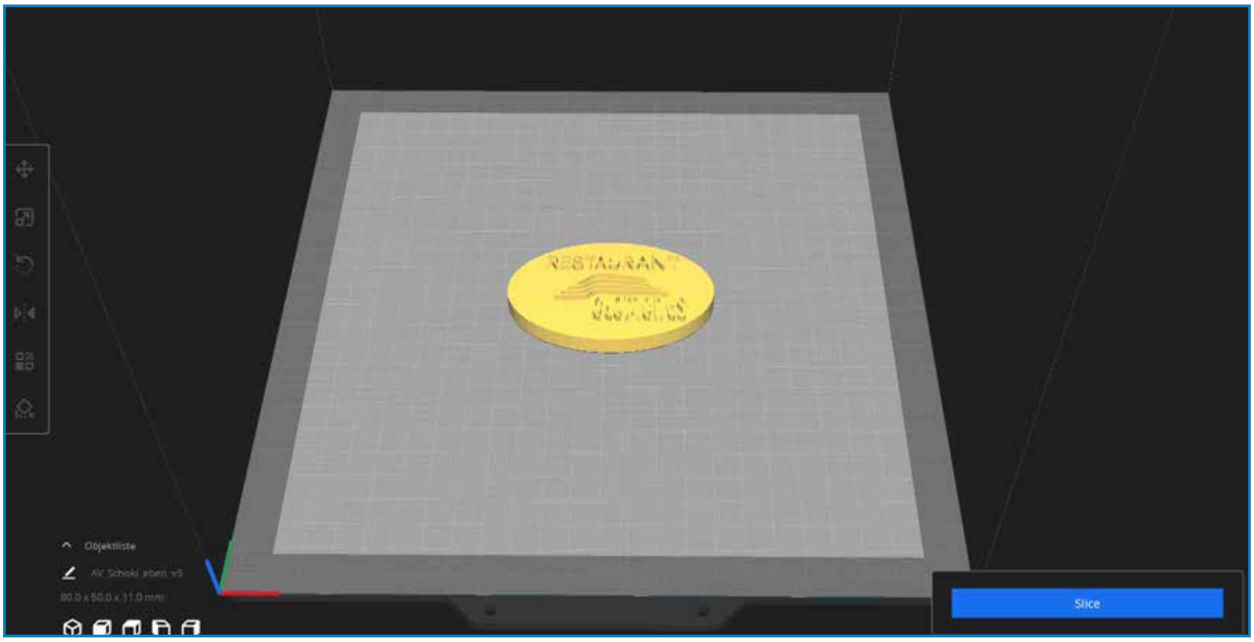
If the product is to have a stretchy property, a TPU material should be used. For high strength, on the other hand, ABS should be used (but only in well-ventilated areas, as toxic gases are released). If the product should be food safe, then PETG would be the right choice. However, PLA is used as the standard filament.

Unit 8

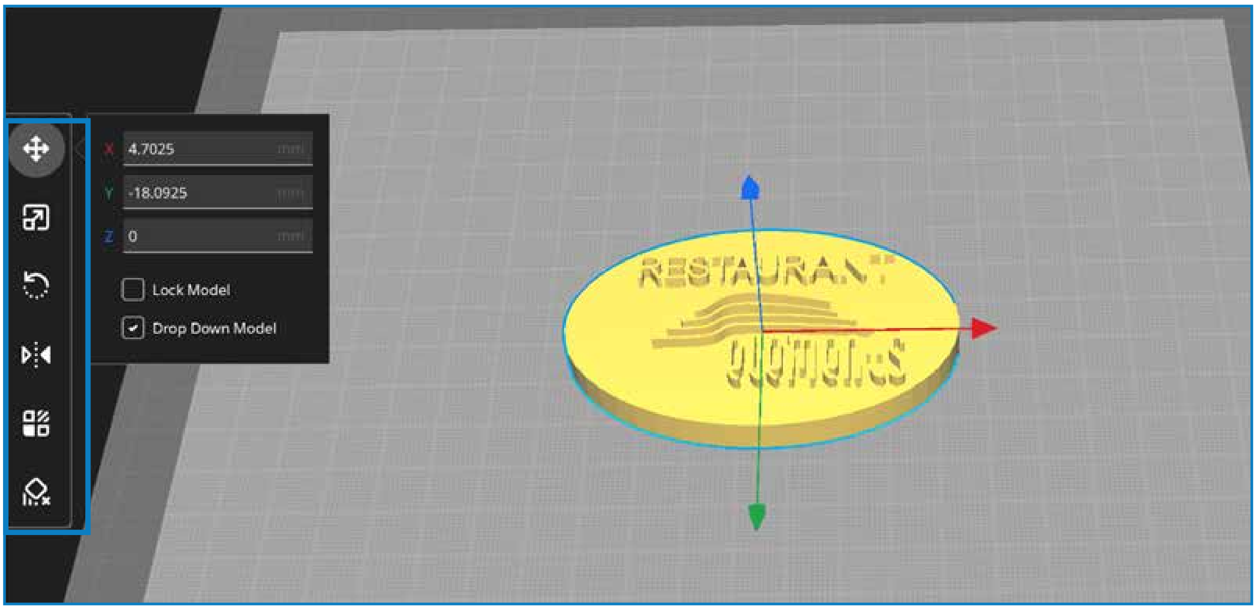
Slice and print model

Once the printer has been calibrated, the desired model can now be prepared in Slicer Cura. It is important that the model is saved as an STL file.

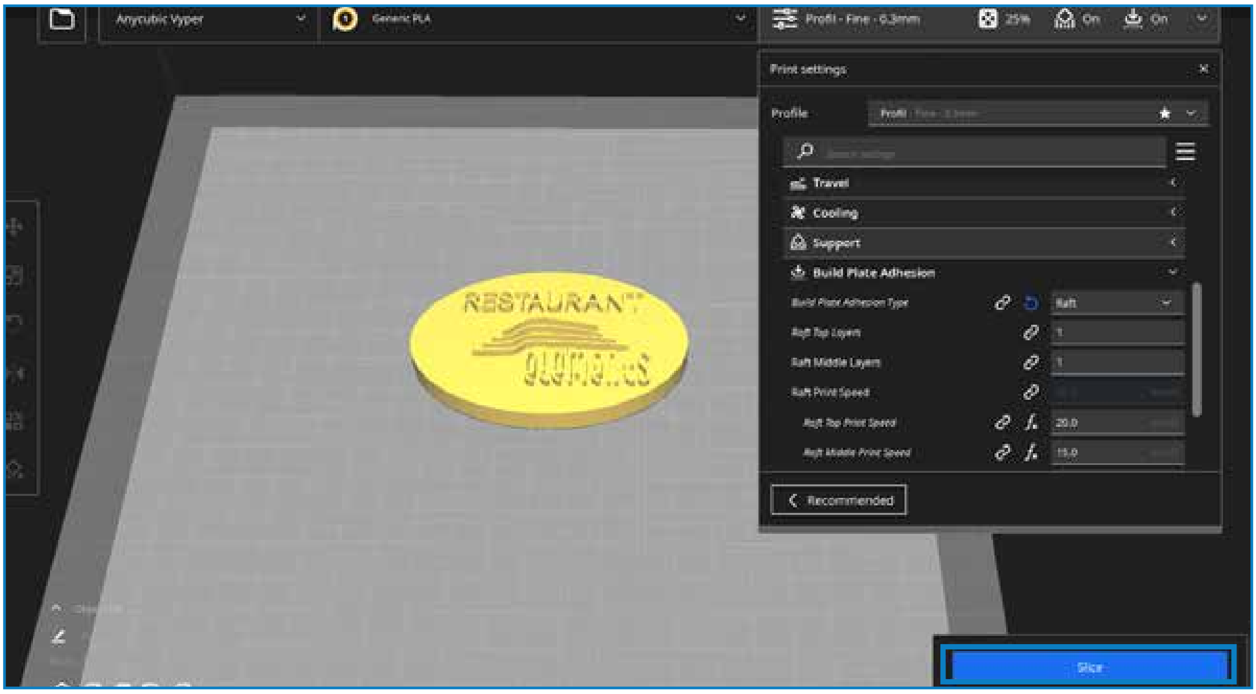
Now open Cura and simply drag the file into the program.



Here the construction is displayed in the original size. The model can be selected by left mouse click. More options should now be available on the left side

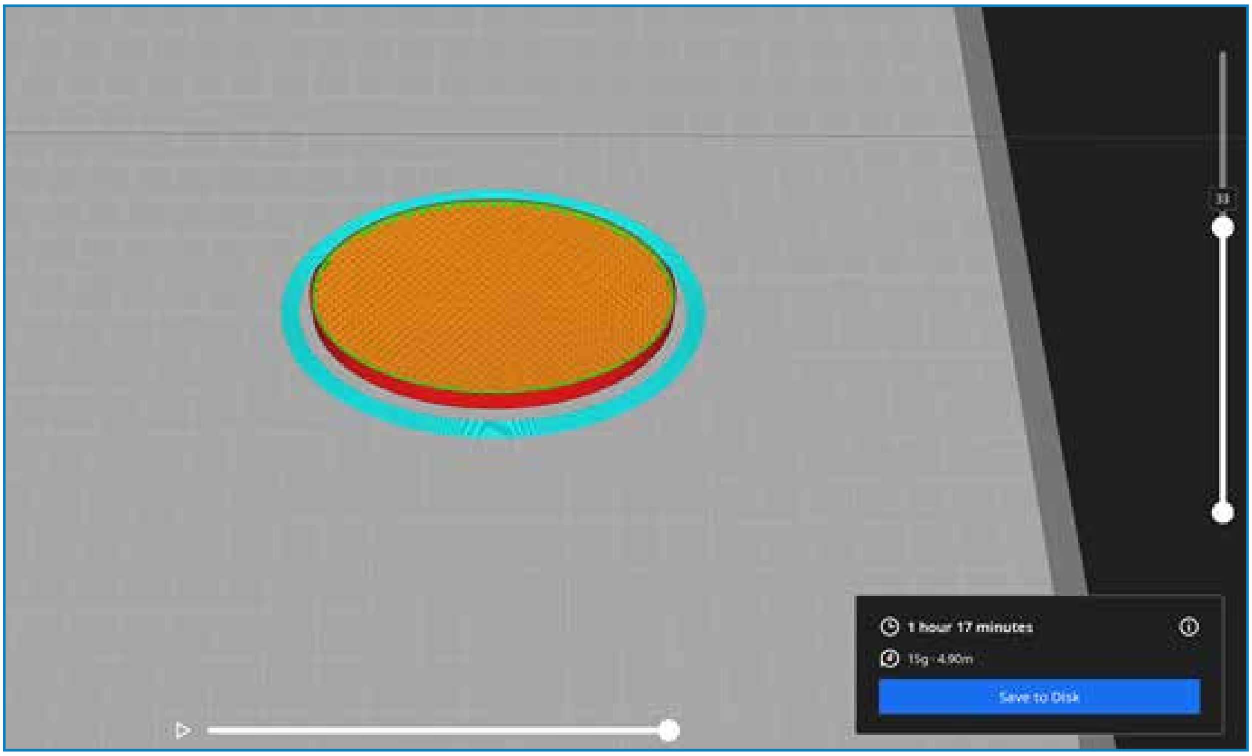


- With the top setting, the model can be moved on the pressure plate.
- With the second setting, the dimensioning can be adjusted. Either all axes can be enlarged/reduced to the same extent or only individual axes. To do this, the check mark at Uniform scaling must be on or off.
- The third option is to rotate the object. Here, too, the model can be moved and optimized in all three axes.
- The fourth setting is mirroring. The model can be mirrored in any direction.



Optimally, the model should always be placed in the centre of the plate. After placing, the individual values of the printer can be adjusted. When this has been done, the model can be sliced at the bottom right.

After the product has been sliced, a preview of the approximate printing time is given, and you get the possibility to view the sliced model layer by layer. For this purpose, the preview option must be selected at the bottom right.



If the settings fit, the model can be saved and transferred to the data carrier.

Now only the data medium must be connected to the printer. Select the model under the printer option Print and start the program.
The printer will then do the rest.

Unit 9

Common printing errors and solutions

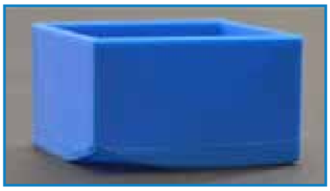
Printing errors with PLA can have various causes. Some of the most common issues include inadequate adhesion to the print bed, too low printing temperature, nozzle clogging, or poor layer adhesion. Uneven filament feeding or inaccurate printer calibration can also lead to errors. It is essential to review the printing settings, ensure the quality of the filament, and, if necessary, calibrate the printer to achieve optimal results with PLA.

9.1 TROUBLESHOOTING WITH PLA MATERIAL

I. First layer does not adhere:

- Pressure plate not levelled properly
- Printing plate not hot enough
- Print head not hot enough
- Printing speed of first layer too low

II. Warping of the model:



- pressure bed too hot
- fluctuating temperature difference in the room

III. Deformation of the model:

- pressure bed too hot
- fluctuating temperature difference in the room

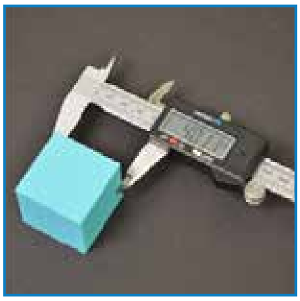
IV. Deformation in thin spots:

- Cooling is too low
- Pressure speed too low

V. Scars on the surface:

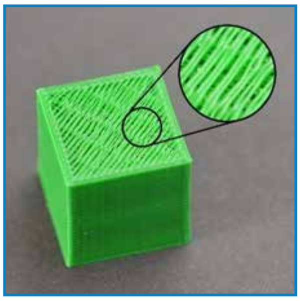
- Z-jump at indentation not on

VI. Gauge blocks not in tolerance:



- Reduce dimensioning
- Increase speed if too small
- Decrease speed if too large

VII. Gaps in the surface:



- Too few surface layers
- too little material is added, adjust feed speed

VIII. Stringing:

- printing speed to slow
- travel speed to slow
- Z-Hop is on

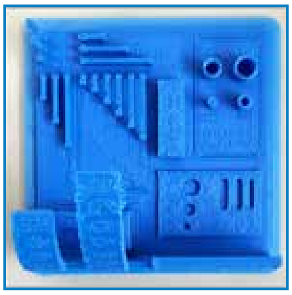
9.2 EXAMPLES

Settings 1

- 0,2-layer high
- 200°C temperature
- 90 mm/sec travel speed
- 40 mm/sec outer wall speed
- 50 mm/sec inner wall speed
- 70 mm/sec filling speed
- combing is activated

Settings 2

- 0,3-layer high
- 190°C temperature
- 60 mm/sec travel speed
- 55 mm/sec outer wall speed
- 70 mm/sec inner wall speed
- 90 mm/sec filling speed
- combing is deactivated



Unit 10

Precision and detail: The world of sla 3D printing

SLA (stereolithography) printing is a 3D printing process in which liquid resin is cured layer by layer using a UV laser. The printing process takes place in a container where the resin is gradually drawn up from a printing plate as the laser cures each layer. When all the layers are cured, the finished 3D model is ready.

✔ **Advantages of SLA printing**	❗ **Disadvantages of SLA printing**
• high detail: SLA printers produce fine and high-resolution prints with a smooth surface and detailed features. • wide range of materials: there are a variety of resin materials for SLA printers, including flexible, transparent, durable and heat-resistant varieties. • No visible layer lines: Because the resin cures in layers, there are no visible layer lines in the final product. • Precise fits: SLA prints have exact dimensions, making them ideal for prototypes and functional models • Fast printing: SLA printers can print relatively quickly, especially for smaller objects.	• Cost: SLA printers and the resin material can be more expensive than FDM printers with PLA filaments. • Material handling: the liquid resin requires special care and should be protected from UV light, otherwise it can harden prematurely. • Post-processing: SLA prints need to be cleaned and post-cured after printing to remove possible residual resin.

Application examples for SLA printing:

1. Jewellery and accessories: Due to the high level of detail, SLA printing is well suited for the production of delicate jewellery and accessories.

2. Prototyping: SLA printing is ideal for producing high precision prototypes in a variety of industries including product design, engineering and healthcare.

3. Dental models: In dentistry, SLA printing is widely used to create precise and accurate models for the production of dentures and orthodontic appliances.

4. Miniatures: SLA printing enables the creation of detailed miniatures for model making or tabletop gaming.

5. Functional prototypes: The precise dimensions and smooth surface make SLA printing suitable for functional prototypes and mechanical components.

The following instructions are intended to help those new to resin 3D printing to gain their first experience. The settings explained here are only rough guidelines and may not apply to every model.

The photos, graphics and screenshots used in this manual are all made by the author, unless otherwise stated.

This manual is based on Anycubic Photon Workshop version 3.1.3 and an Anycubic Mono X with firmware version 3.5.4. Newer and older versions may differ in function and appearance.

10.1 SYSTEM REQUIREMENTS

Anycubic recommends end devices with the following system parameters for the use of Photon Workshop (version 3 and higher):

Windows operating system	**MacOS operating system**
• CPU: Intel® Core™ i5 6600K or better / AMD Ryzen™ 5 1600 or better • RAM: ≥ 16 GB • HDD: 2 GB • Screen resolution: min. 1920x1080 / recommended 2560x1440 or better • GPU: NVIDIA GeForce GTX1050 or better / AMD Radeon RX480 or better • Graphics memory: ≥ 1 GB	• CPU: Intel® 4-Core (OS-Version 10.15 or better) / Apple M1 4-Core (OS-Version 13.0 or better) • RAM: ≥ 16 GB • HDD: ≥ 64 GB • Screen resolution: min 2560x1440 or better

10.2 SOFTWARE DOWNLOAD

The current version of the Photon Workshop can be downloaded at

https://www.anycubic.com/pages/anycubic-photon-workshop-3d-slicer-software

The following QR-Code also leads to the download page, where you can also find the latest manuals.

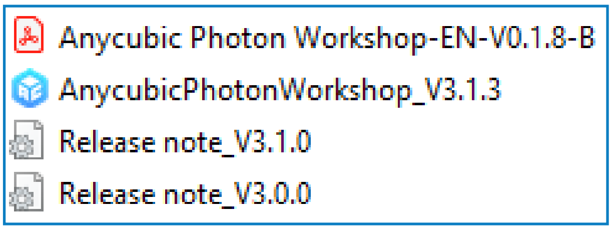
On the download page you can choose between the operating systems Windows and MacOS and the respective versions of Photon Workshop. It is generally recommended to always use the latest version. However, if problems occur with this version, the next older version should be selected. When using older devices, it may be necessary to perform several downloads until a version is installed that will run on the device in use.

The download is a .zip file.

10.2 INSTALLATION

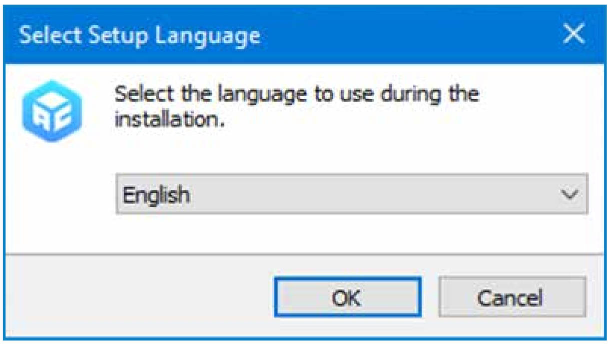
To install Photon Workshop, first unzip the downloaded .zip file. A new folder is created with an install file, the current user manual and the release notes.

The install file is used for the installation itself.

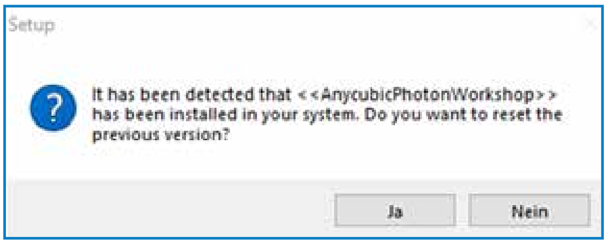


In the first step, the installation file asks you to select a system language. In the present version, you can choose between English and Chinese via a drop-down menu.

The selection of the desired language is confirmed with „OK“. In the next step, the setup wizard opens and guides you through the installation in the selected language.

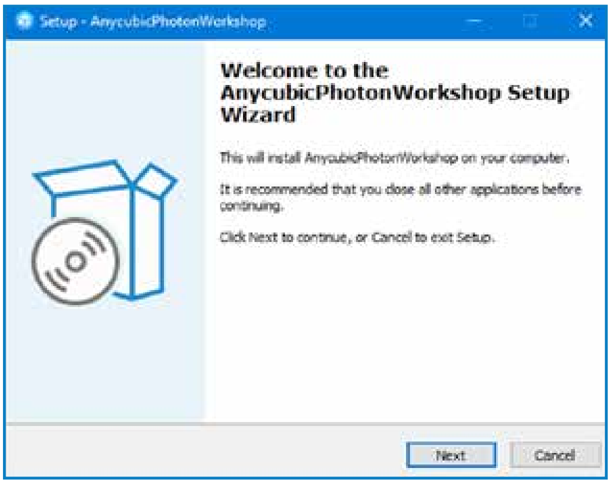


If there is already an older version of Photon Workshop installed on your computer, you will be asked if you want to overwrite this version.

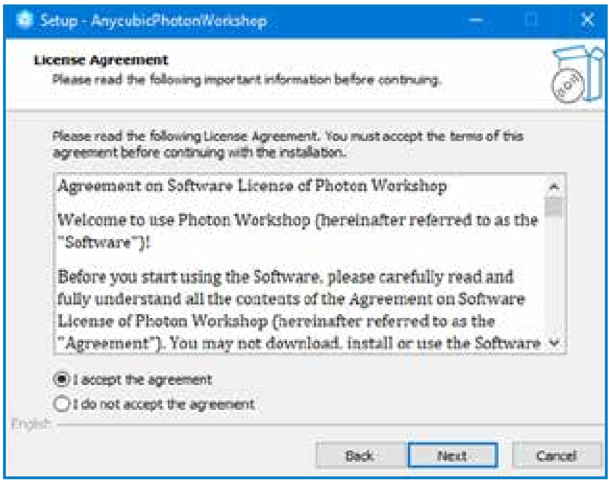


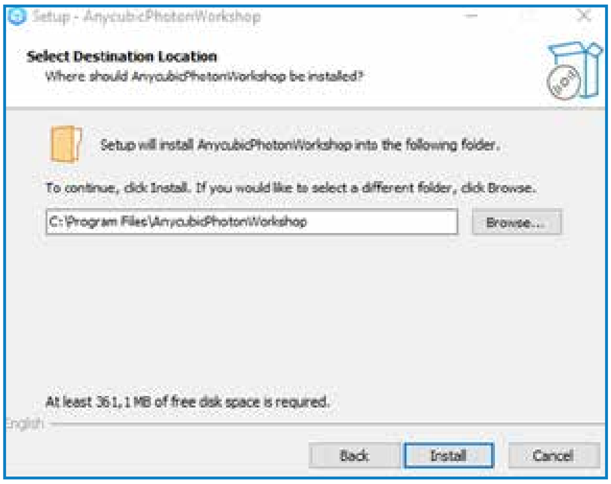
Attention

It is possible that parameters set in the older version, such as printers and resins, will NOT be imported into the newer version. To avoid loss of data, it is recommended that the manually set parameters are stored externally. This is explained in more detail in a later chapter.



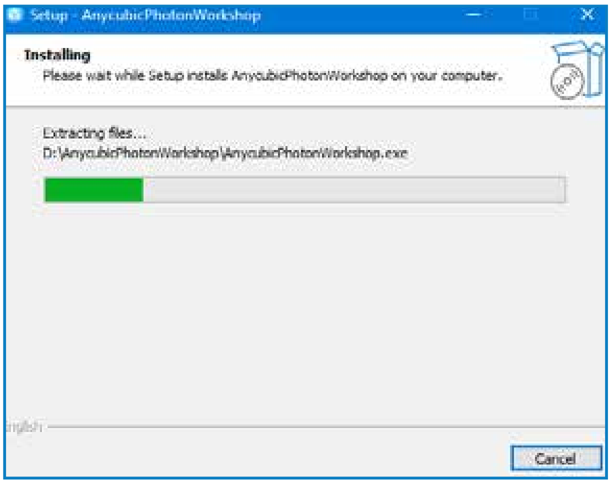
By clicking on „Next“, the next step is the software licence agreement. This must be accepted in order to carry out the installation.





Now select the target location for the installation. The setup wizard automatically suggests the main memory as the installation location. Another storage location can be selected via „Browse...“.

Select ‚Install‘ to start the Photon Workshop installation.



The completion of the installation is confirmed by clicking on „Finish“. Photon Workshop can then be opened via the desktop icon or via the installed folder.

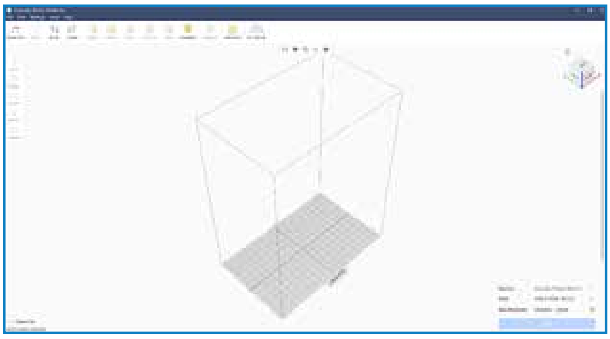
Unit 11

Start and basic settings

11.1 HOW TO START PHOTON WORKSHOP

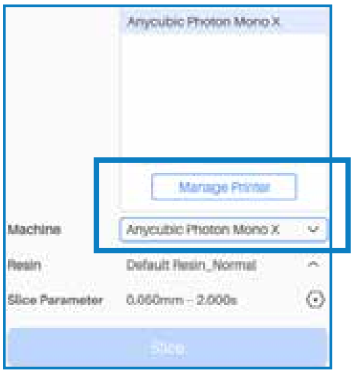
To start Photon Workshop, click on the corresponding desktop icon or the icon in the installation folder using the Explorer.

If successful, the loading screen will appear.



Once fully loaded, the Photon Workshop main screen appears.

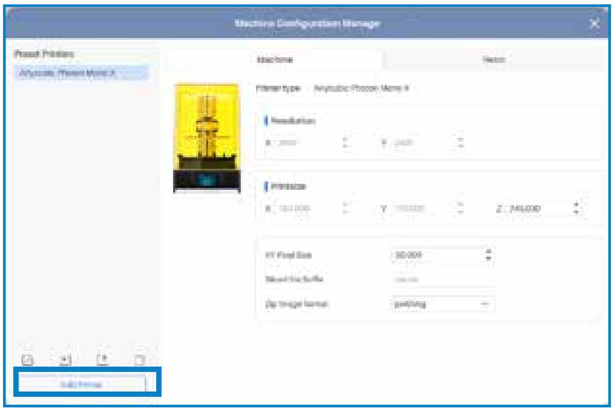
11.2 BASIC SETTINGS - DEVICE PARAMETER



Start by configuring your printer type. Open the printer setup menu by clicking on the ‚Machine' drop-down menu in the lower right corner.

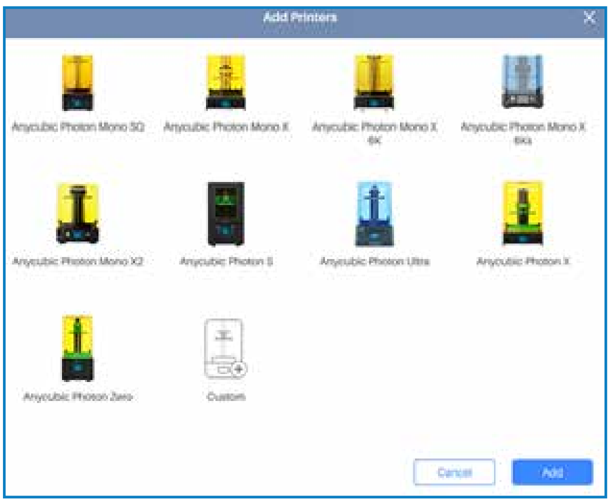
Pre-set printers are directly shown in the list.

New printers can be added via the tab “Manage Printers”.

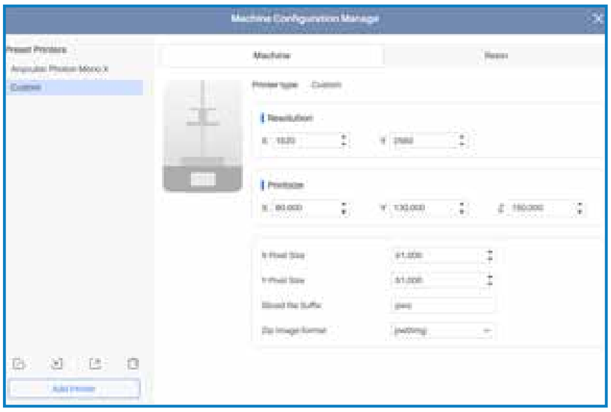


The „Machine Configuration Manager“ displays all machine parameters relevant for printing. The item „Resolution“ shows the maximum possible number of pixels in X and Y direction. In the example of the Photon Mono X printer shown, this would be 3,640 pixels in the X direction and 2,400 pixels in the Y direction, i.e. a total resolution of 3,640 x 2,400 pixels. The print size indicates the maximum size of the build platform and the maximum print height (in the example: 192x120 mm). With pre-set printers, these values cannot be changed, only the Z-direction, i.e. the print height, can be changed. However, it is only recommended to deviate from the pre-set parameters if a smooth functioning of the spindle in a certain area (e.g. in case of damage in the upper spindle area) can no longer be guaranteed.

To add a new printer, select the „Add Printer“ button.



All printers of the manufacturer Anycubic are already preset in the menu for „Add Printer“. If no Anycubic printer is used, select the option „Custom“ and confirm the selection with „Add“.



The selected printer will now appear in the left-hand drop-down list. You can now adjust the previously mentioned parameters, such as ‚Resolution‘ or ‚Print Size', according to the properties of your selected printer. If the printer name is displayed as ‚Custom', you can change it by clicking on the ‚Rename‘ button in the bottom left hand corner.

11.3 BASIC SET-UP OF THE RESIN PROPERTIES

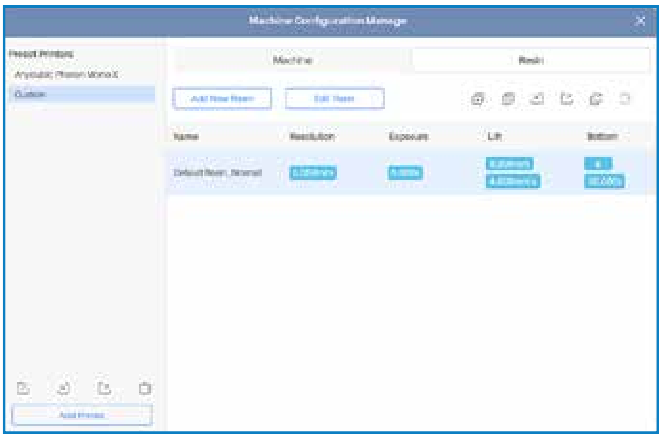
Resins are high quality casting resins obtained by dehydrating natural resins. Their key feature for 3D printing is their photopolymerisation (light-sensitive). These properties allow the resins to cure precisely and in a matter of (fractions of a) second(s).

The applications for resins are extremely varied:

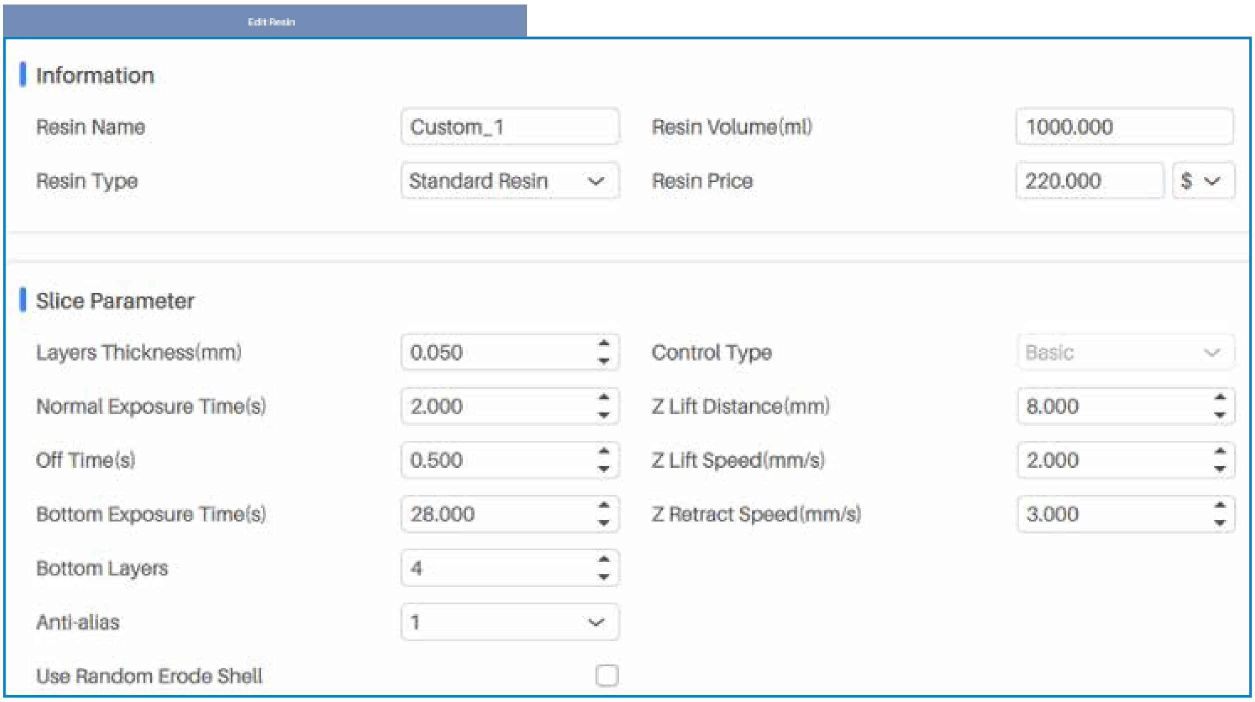
- Standard resin: ideal for leisure, hobby and school use.
- Craftsman resin: perfect for crafts and model/prototype building where strength is not required.
- ESD resin: specifically for use in microelectronics to prevent electrostatic discharge.
- Dental resin: used in medical technology for dental applications.

Resins are also available in a wide range of colours, including translucent clear variants. Note, however, that the colour pigments used can affect the photopolymer properties, particularly the exposure and curing times required.

In the Photon Workshop software, you can create new resins and manage existing ones under the „Resin“ tab, similar to the „Machine“ tab. Here you have full control over the different material options for your SLA printing process.



Use the button “Add New Resin” to create new resins and “Edit Resin” to set the parameters for printing with this resin.



Under ‚Information‘, general information about the resin used can be entered, such as the name, type of resin or price per litre (used later for a system cost calculation).

Under „Slice Parameters“ you can enter parameters that are important for the printing result.

Control Type

Under Control Type, you can choose between two slice parameter settings, Basic and Advanced. If you are new to resin printing, the ‚Basic‘ setting is recommended. In these settings, as shown above, only the rough parameters for slicing are entered. This means that the software itself calculates and sets the parameters for how the printer will move. The ‚Advanced‘ setting allows the user to define the parameters for the printer‘s movement.

Layer Thickness (mm)

The layer thickness of a single layer in the Z-direction is specified here. The finer the layer thickness, the more accurate the print in the Z direction. It should also be noted that the thicker the layer, the longer the exposure time must be.

Normal Exposure Time (s)

The exposure time for a normal print layer can be set here. The time usually depends on the thickness of the layer, the power of the UV lamp of the adapted printer and the complexity of the model to be printed. If the curing time is too short, parts of the model may not cure properly, resulting in cracking during printing or even half-dried particles remaining in the resin. If the curing time is too long, resin may be left on the edges of the model where it should not have dried, and the final print will be inaccurate.

Off Time (s)

Describes the time during which the model rests in the printing position before the UV lighting is switched on again. With viscous resins it is recommended to increase the resting time in order to obtain an even print image.

Bottom Exposure Time (s)

Describes the exposure time of the first layers of the print. The longer the exposure time, the better the model will adhere to the platform. But beware, the better the adhesion to the base plate, the harder it is to remove the model after printing.

Bottom Layers

These are the base layers on the building platform and are usually larger than the model. It is recommended to print several base layers on top of each other, depending on the model. Roughly speaking, the heavier the finished model, the more base plates are needed.

Z Lift Distance (mm)

Specifies the distance the Z-axis should be moved upwards to detach the model from the FEP film.

Z Lift Speed (mm/s)

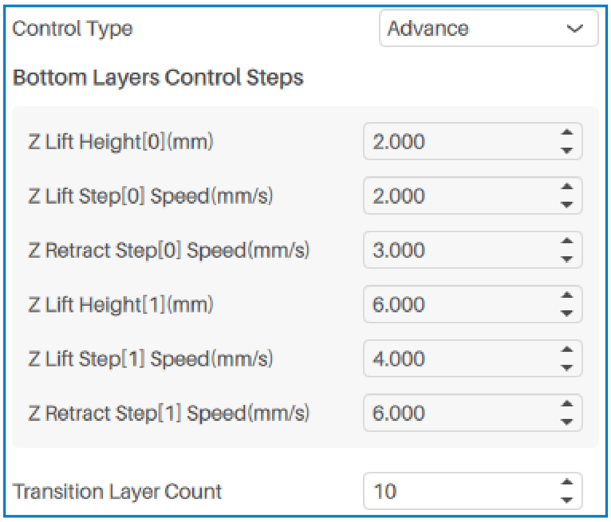
Specifies the speed at which the Z-axis should move upwards. Caution: If the speed is too high, the printed model may be damaged or the last cured layer may peel off the model and remain on the FEP film.

Z Retract Speed (mm/s)

Specifies the return speed of the Z-axis to the next print position. The speed should not be set too high, as the liquid resin bath may not be able to settle within the off time and the quality of the print may deteriorate or become distorted.

Control Type „Advanced“

The „Advanced“ control mode provides additional setting options. These are mainly used to optimise the printing process, both in terms of reducing printing time and increasing process reliability.



In the „Advanced“ mode, the previously mentioned setting options are differentiated. In particular, a distinction is made between the movement speeds near the layer to be printed („Step[0]“) and at a certain distance from the layer to be printed („Step[1]“). In summary, one can say that in the area of „Step[0]“ low speeds should be run in order not to negatively influence the print image, and in the area of „Step[1]“ higher speeds should be run in order to reduce the total printing time.
Even more important, however, are the listed “Transition Layers“.

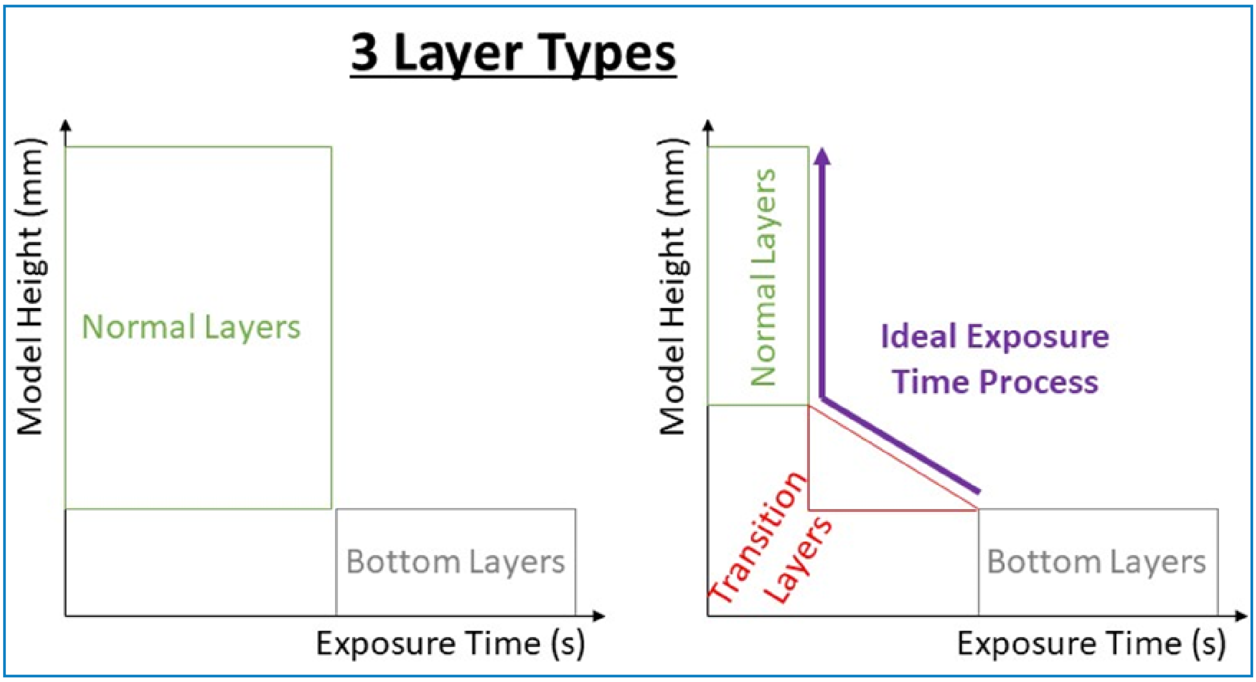
Transition Layer Count

Transition layers are, as the name suggests, intermediate layers between the „bottom layer“ and the „normal layer“ in relation to the exposure time set.

If the exposure time is reduced too quickly (in extreme cases even within the same layer), e.g. from 45 seconds (bottom layer) to 4 seconds (normal layer), problems can occur. The printed model may crack, layers may peel off or warp. In the worst case, the cured layer will stick to the FEP film because the adhesive forces on the printed part are lower than on the film.

To avoid these problems, the exposure time of the „transition layer“ is gradually reduced until it reaches the exposure time of the „normal layer“.

This allows a smooth transition between layers and helps to keep the printed model stable and accurate.

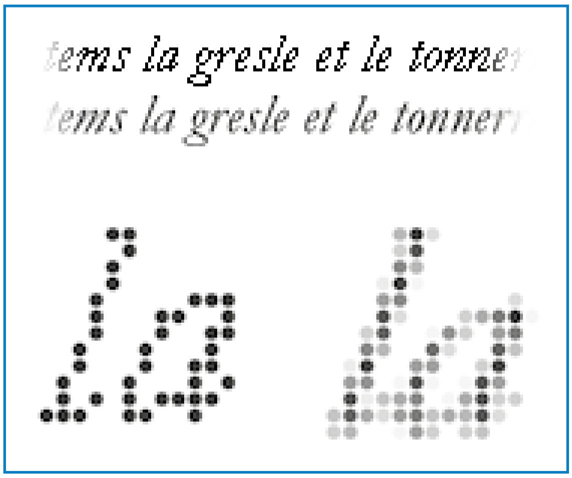


The number of transition layers depends on the exposure time reduction from the bottom layers to the normal layers. In „Basic“ mode, this reduction is set by the software and is usually ten transition layers. The software only deviates from this rule if the reduction would be more than 6 s per layer. This rule of thumb that the maximum reduction is 6 s per layer can be applied to the majority of resins.

Model pre-processing

Anti Alias

Antialiasing (edge smoothing) is the reduction of unwanted effects that can occur due to the limited pixel grid of the UV screen (see point 2.2.1.). In the three-dimensional area, the staircase effect can occur in particular. The staircase effect is the sometimes „staircase-like“ appearance at the edges of figures (see example below). The trigger for the staircase effect is simply the finite number of pixels available in the UV screen.



rasterised text without (left) and with Antialiasing (right)

In resin printing, the UV screen only knows black and white, similar to the left part of the picture. In order to minimise the staircase-like shape of the print, the model is smoothed by so-called antialiasing. In this process, the areas shown in grey in the image are also illuminated by the UV screen. The higher the antialiasing, the smaller the staircase effect. However, this leads to a longer calculation time during the slicing process and to a larger file size of the finished slice.

Use Random Erode Shell

This option is only possible with an antialiasing value of 1. With this option, the staircase effect is specifically used to create a „frayed“ and thus matt appearing surface.

Grey Level

This option is only possible with an antialiasing value of 2 or higher. In this case, the greyscales in the antialiasing are more finely divided in order to achieve a higher surface quality in the finished model. However, this also leads to a larger file size and a higher slice time.

Image Blur

This option is only possible with an antialiasing value of 2 or higher. The edges of the model are artificially blurred further. This leads to a higher surface quality, but also to a larger file size and a higher slice time.

11.4 SETUP OF A RESIGN

Below you will find the setup of a new resin using the example of „ANYCUBIC Basic Resin“ in grey colour with the guiding questions „Where can I find suggested parameters for the respective resin?“ and „Which parameters do I use in the settings?”

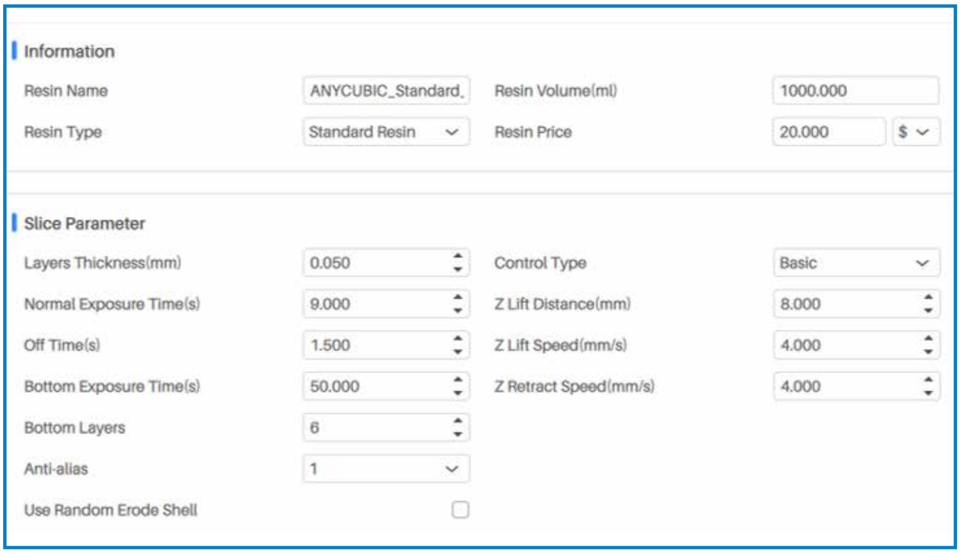
Suggested manufacturer settings

Resin manufacturers usually give rough suggestions for the basic settings. These can usually be found directly on the resin bottle, on the packaging or in the product information sheet. Many manufacturers now also print QR codes on the bottles and/or packaging. These then lead to the homepages of the manufacturers, where the suggestions are often also noted.

The picture shows the back of the resin bottle. ANYCUBIC suggests an exposure time for the „Bottom Layer" of 20-80 s and for the „Normal Layer" of 3-15 s.

Start parameters for the print

ANYCUBIC suggests parameters with a relatively wide range for printing. It is recommended to start with the middle of the suggested parameters, i.e. with an exposure time for the „bottom layer" of 50s and the „normal layer" of 9s.



The picture shows the current parameters. When working with new, unknown resins, it is advisable to increase the „Off Time" somewhat, as the flow properties of the resin are not yet known. A good starting value for the „Z Lift Distance" is 8 mm, because at this height it can be safely assumed that the printed model is definitely separated from the FEP foil or in the worst case from the base plate.

The „Z Lift Speed" and „Z Retract Speed" of 4 mm/s are also good starting values, since both the detachment behaviour of the model from the FEP film and the flow behaviour of the resin are not yet known. After a successful print, these parameters can be further refined. In particular, exposure times can be reduced and Z-axis movement speeds increased.

Steps of 5 seconds are recommended for the „bottom layer" and 0.5 seconds for the „normal layer". Steps of 1.0 mm are appropriate for the Z-axis movement length and steps of 0.5 mm/s for the movement speed. If the first print is not successful, the parameters should be adjusted in the same steps in the other direction.

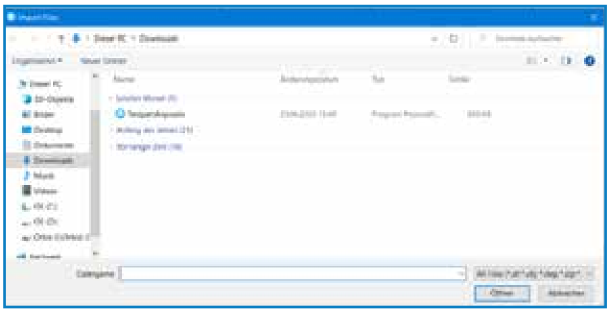
Unit 12

Creating slices

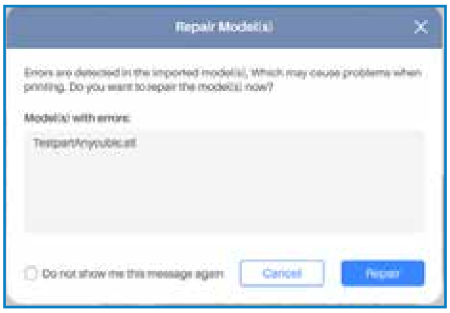
The following chapter explains in detail how to create the slice files needed to operate the printer. It explains which software and settings are used to convert the 3D model into layer-by-layer information for the printer. The slice files contain all the necessary instructions for the printing process, including exposure times, movement of the printing plate and other parameters.

It is important to create the slice files carefully as they have a direct impact on the quality of the printed model. An accurate and well-configured slice file will result in a successful print with smooth surfaces, clear details and high accuracy. It is therefore advisable to familiarise yourself with the various settings and options in order to achieve the best possible result.

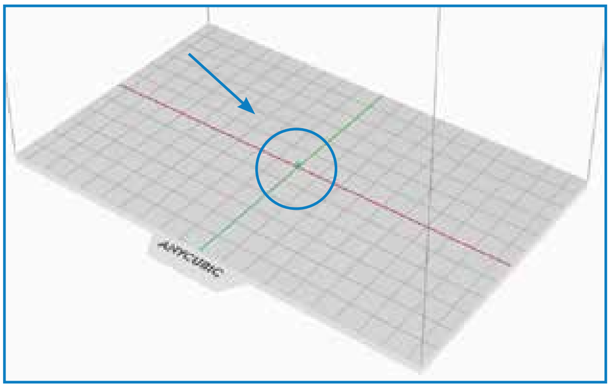
12.1 LOADING OF 3D MODELS



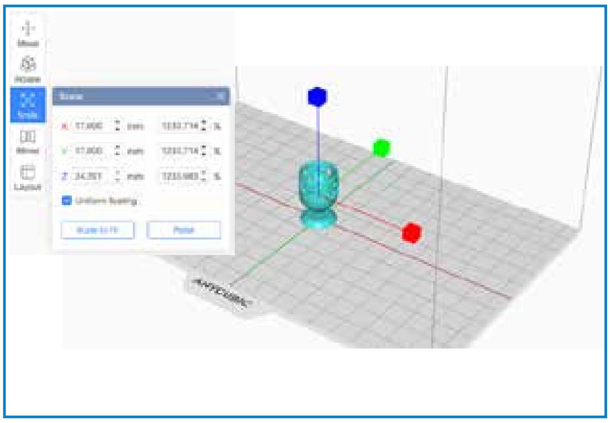
Created 3D models can be loaded either via the „Open File“ button or the shortcut „Ctrl + O“. The desired file can be selected in the dialogue window and loaded into Photon Workshop via the „Open“ button.



When converting 3D models from CAD programs to .stl, .step or .iges format, small errors can occasionally occur, such as polygons that are not closed. Photon Workshop detects these errors and offers to repair them. Pressing the „Repair“ button will repair the file and load it into the workspace.



Due to the repair, the model (turquoise dot in the picture) may be smaller than originally designed. In this example, an egg cup was designed with a planned diameter of 34mm at the base. However, it is currently only 2.75mm in diameter. To scale the model back to its original size, you can either left-click on it or select it from the object list at the bottom left of the screen. You can then use the „Scale" menu item on the left to scale the model to the desired size.



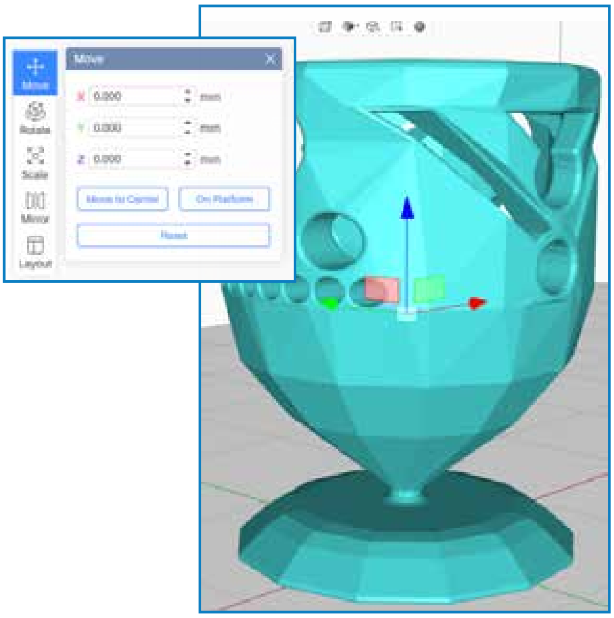
After entering the required dimensions in the appropriate direction of the coordinate system, press ‚Enter'. It is important to ensure that the ‚Uniform Scaling' command is activated, otherwise the values for each axis must be entered individually. The ‚Scale to fit' command would scale the model to the maximum possible printing size. The ‚Reset' button is used to reset the scaling changes.

12.2 MOVING THE MODEL ON THE BUILDING PLATFORM

Photon Workshop always places the loaded 3D model in the centre of the building platform. If several models are loaded into the software in succession, they are automatically placed one inside the other. However, this makes printing difficult. In the menu bar there are several options to move the models on the workspace.

12.3 MOVING THE MODELS ON THE AXIS ON THE COORDINATE SYSTEM

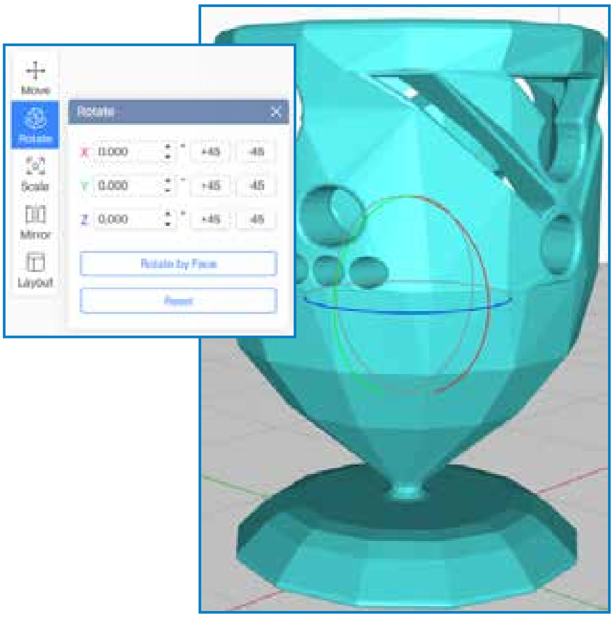
The simplest movement is to move the model along the X and Y axes. This is done by gripping the model with the left mouse button. With the left mouse button held down, the model can be moved in the horizontal plane of the building platform.



The ‚Move' menu allows you to move the selected component along the appropriate axes using the ‚Arrow' buttons or by directly entering the offset values in the corresponding axis. You can also select the peaks of the displayed coordinate system in the model and move the model along the selected axis by holding down the left mouse button.

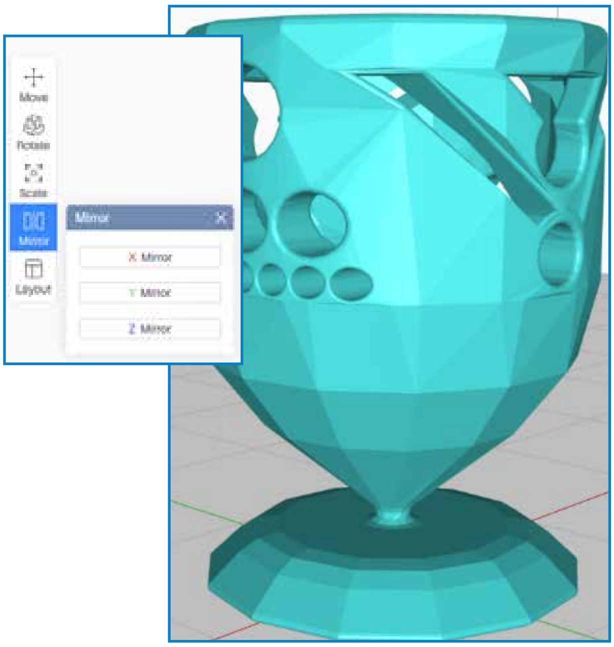
The button „Move to Center" returns the model to the origin of the coordinate system, i.e. the centre of the building platform. The button „On Platform" sets the model to the building platform, i.e. to Z=0.000. With the button „Reset" all changes made can be undone to restore the original state.

12.4 ROTATING THE MODELS ABOUT THE AXES OF THE COORDINATE SYSTEM



Via the „Rotate" menu you can rotate the selected component along the respective axes. This can be done either via the „Arrow" buttons or by directly entering the rotation values in the corresponding axis. To simplify matters, there are also preset buttons with +/- 45° that enable a quick rotation. In addition, you can select the circular paths of the displayed coordinate system in the model and, while holding down the left mouse button, rotate the model along the selected axis.

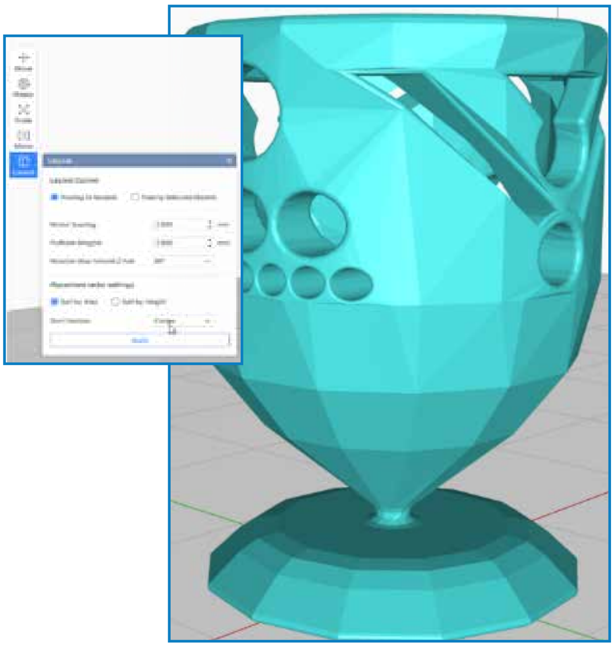
The „Rotate to Face" button allows you to select a face of the model to rest on the build platform. The system performs the necessary rotational movements of the model directly to achieve this. With the „Reset" button you can undo all the changes you have made and restore the original state.



Using the Photon Workshop programme, you can mirror the selected component along the respective axes. This allows you to mirror the model horizontally or vertically, depending on which axis is selected.

The Mirror function provides an easy way to duplicate the part or adjust the print orientation. For example, if you want to print a symmetrical model, you can mirror it along one axis to save time and material.

To mirror the model, you can either use the „Arrow“ buttons in the „Mirror“ menu or enter the desired values directly. Preset buttons for mirroring by +/- 45° are also available here to make the process easier.



If several models are to be printed at the same time, the Layout menu allows you to automatically set the position of each model on the building platform.

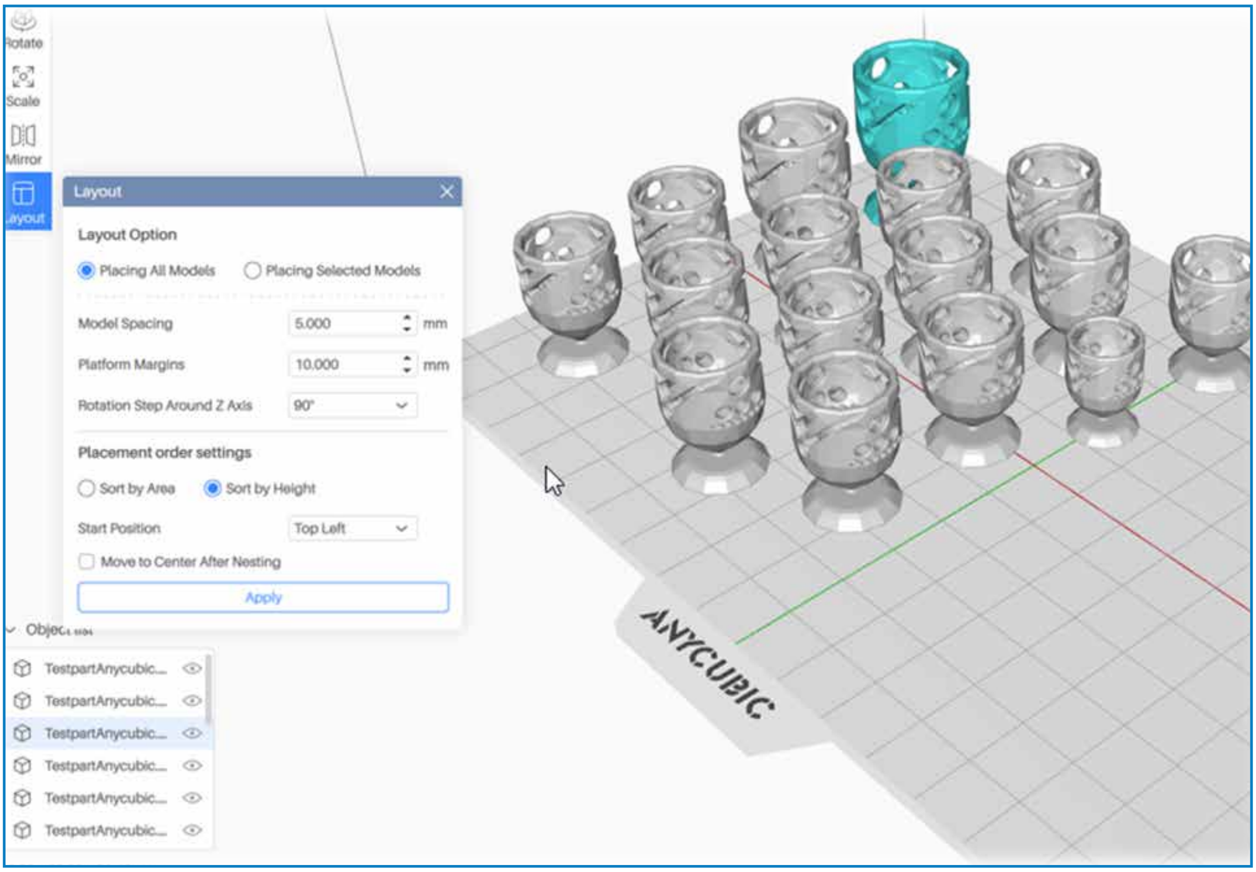
First select whether only the models selected from the „Object list“ or all models are to be placed. „Model Spacing“ specifies the minimum distance between the models, while „Platform Margins“ specifies the distance of the models from the edges of the building platform. „Rotation Step Around Z Axis“ limits the angle by which the models can be tilted during placement.

The next step is to select how the models are to be sorted on the building platform. The option „Sort by Area“ places the models based on the size of their footprint, while „Sort by Height“ takes into account the size along the Z-axis.

Finally, the starting position for the positioning is chosen. This can start either from the centre or from one of the corners of the building platform. By pressing the „Apply“ button, the placement is started and the models are automatically arranged on the building platform according to the selected settings.

The following image shows a full building platform with several egg cups as an example. These were scaled differently for illustration and placed with the following parameters:

- „Placing all Models“
- „Model Spacing“: 5mm
- „Platform Margins“: 10mm
- „Rotation Step Z Axis“: 90°
- „Sort by Height“
- „Start Position“: „Top Left“



From the picture you can see that Photon Workshop starts with the largest egg cup at the starting position.

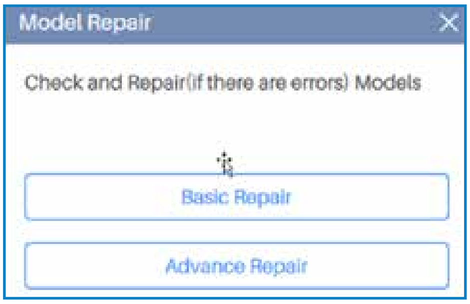
Attention:
It is recommended to vary the printing position of the models with each print in order to avoid punctual wear of the FEP foil and the building platform.

12.5 MODIFICATION OF 3D MODELS IN PHOTON WORKSHOP

In Photon Workshop, 3D models can be fundamentally modified. This is mainly done via the upper menu bar, as shown in the following example.

12.6 REPAIR OF DAMAGED MODELS

As already mentioned in chapter 13.1 Loading 3D models, Photon Workshop offers tools for repairing 3D models that are unsuitable for printing or damaged.

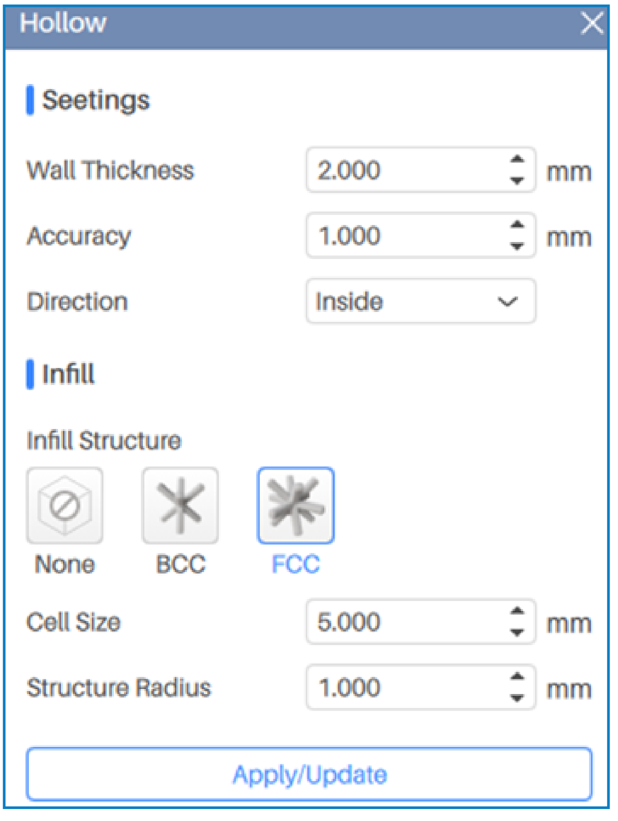


The ‚Repair' button allows you to choose between two repair methods. „Basic Repair is the simplest and quickest way to make a model printable. Missing or overlapping polygons are closed or removed.

With Advance Repair, the problem areas are marked and separate actions must be selected for each area. Advance Repair is recommended if the models to be printed have been created using sculpting methods (e.g. Blender software) and often have more complex and finer polygon meshes.

12.7 HOLLOWING OUT THE MODELS

In resin printing, models are printed as solid material in the basic setting. However, Photon Workshop offers the option of hollowing out these models and creating an infill structure, similar to the FDM printing processes. This is particularly advantageous for larger and tall models that are not exposed to high mechanical stress. By using infill, material can be saved, resulting in lower printing costs and shorter printing times. However, it should be noted that the print stability and strength of the model can be affected by the voids. It is therefore advisable to choose the optimal infill density and structure depending on the application and the model.



To print hollow models using resin printing, the „Wall thickness“ and „Accuracy“ (wall thickness tolerance) must be set in the menu. These parameters depend on the overall structure and volume of the model and must be selected accordingly. It may take several trials to find the right settings for correct printing. The direction of the voids can be selected in the „Direction“ menu, usually they are placed inside the model. The „Infill Structure“ of the cavity is then defined in more detail. Photon Workshop offers three choices:

- „None“: No filling
- „BCC“: Creation of a support structure in the form of a coordinate system inclined by approx. 30°
- „FCC“: Creation of a support structure in the form of a coordinate system inclined by approx. 30° and additional supports at a 45° angle starting from the centre of the inclined coordinate system

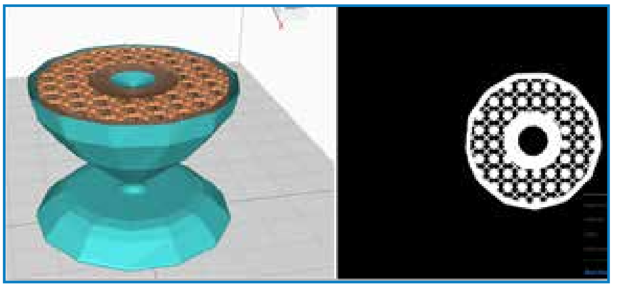
The choice of support structure should be based on the expected mechanical stresses of the printed model. In general, the more complex the support structure, the better the model can absorb possible mechanical loads.

The size of a support structure unit „Cell Size“ and the diameter „Structure Radius“ of the individual arms of the support structure also influence the cushioning.

The following picture shows an example of a hollowed out egg cup with a support structure in section. For clarity, the egg cup has also been enlarged.

The parameters for hollowing and generating the support structure are as follows:

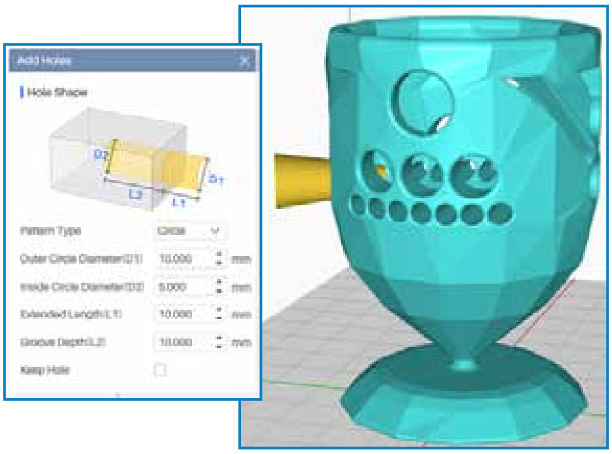
- Wall Thickness“: 2mm
- „Accuracy“: 1mm
- „Direction“: „Inside“
- „Infill Structure“: „FCC“
- „Cell Size“: 5mm
- „Structure Radius“: 1mm



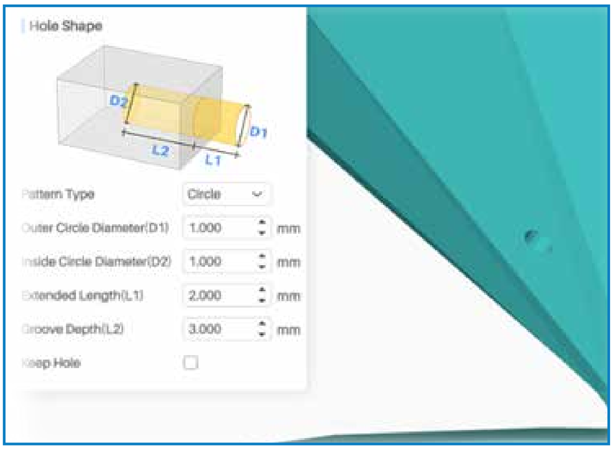
The left side of the image shows the section in perspective view, the right side shows the section in plan view.

12.8 ADDING FURTHER APERTURES

Photon Workshop offers the user the possibility of subsequently inserting further apertures into the model using the „Punch“ function.



The apertures are defined in their geometric form using the „Punch“ menu. They can then be placed freely on the model using „Point and Click“. In the current version, only circular basic shapes can be defined („Pattern Type - Circle“). These can also be made conical by varying the inside and outside diameters and the inside and outside lengths.

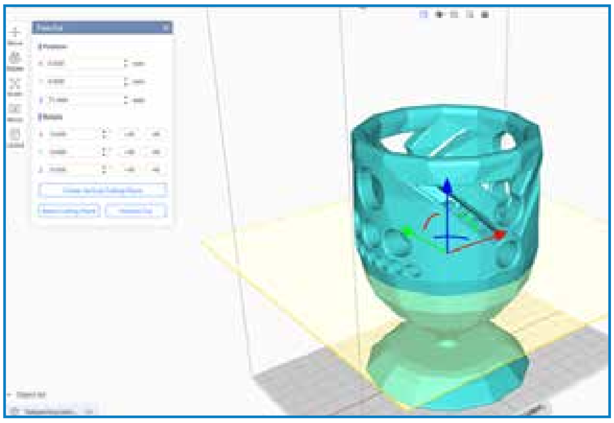


The main purpose of these openings is to allow uncured resin to flow out of a hollow model, as explained in the previous chapter, and to avoid air pockets that can permanently damage the print.

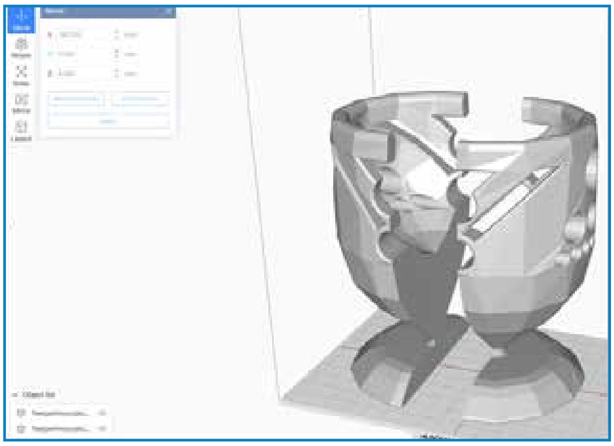
In the example of the egg cup, a small opening just above the cone would be advantageous, as shown in the following picture.

12.9 CUTTING THE MODEL

Photon Workshop offers the user the possibility to split models into two independent models using the „Free Cut“ function.



In the menu for „Free Cut“, the position of the cutting plane can be moved in relation to the coordinate system and its position can be changed by rotating it around the respective coordinate axis. The „Create Vertical Cutting Plane“ button can be used to draw a new cutting plane manually with the cursor. The button „Reset Cutting Plane“ resets the cutting planes to the original state. The „Perform Cut“ button executes the cut.



As shown in the illustration, the cut creates two independent models, which are also kept separate in the „Object List“. The already cut models can then be split again as often as desired.

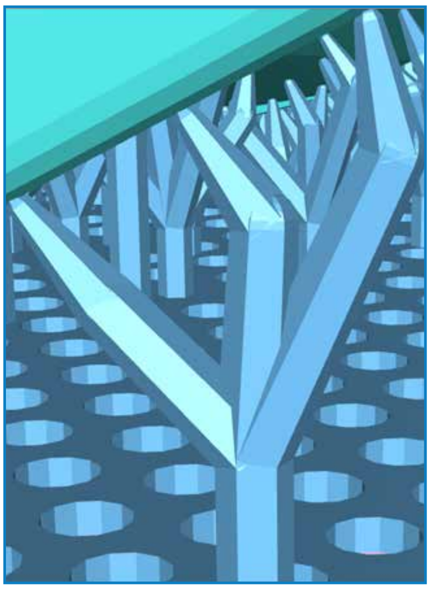
12.10 ADDING LABELS

Adding text is done using the +Add tab. Enter the text you wish to add. Then use the cursor to select the area where you want the text to be placed. The font and its size can be selected using the ‚Font‘ and ‚Font Size‘ tabs. „Font Height“ determines how far the font will protrude from the surface of the model. Rotation allows you to change the angle of the font along the selected surface. Once you have made all the settings, confirm them with the „Apply“ button and the label will be created.

The tab „-Delete“ can be used to remove individual selected labels or all texts attached together.

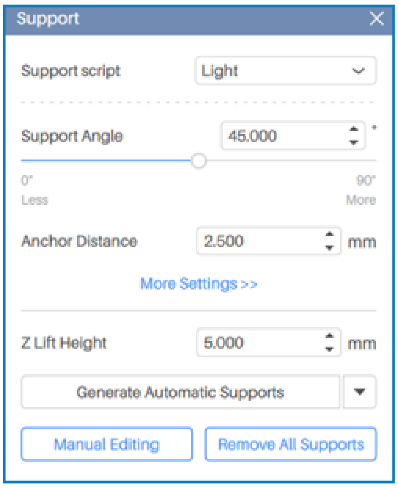
12.11 CREATING EXTERNAL SUPPORT STRUCTURES

For heavier models, models with overhangs of more than 50°, or models with severe necking, as in the egg cup example, it is necessary to attach external support structures to the model, otherwise the models may be damaged (e.g. warped) or cracked during printing. These can be created using the “Support” function.



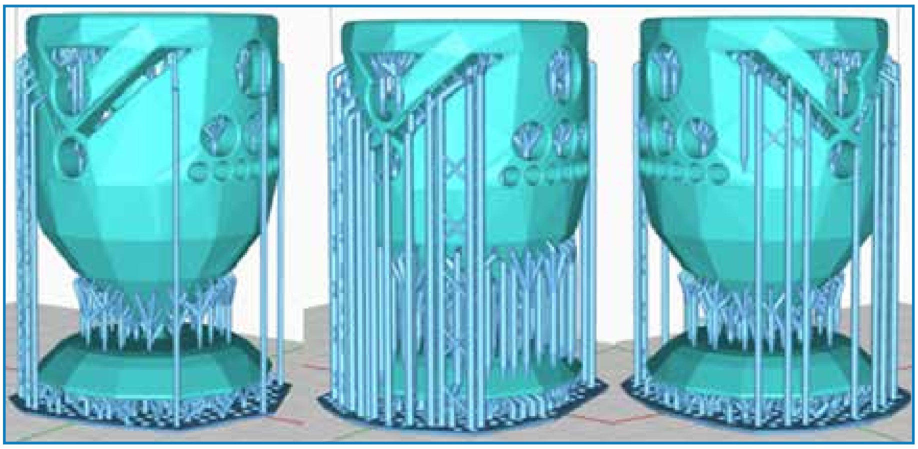
The different parts of the support structure can be compared to the structure of a tree. At the bottom is the strong or thick trunk of the tree. Further up this is divided into a number of branches from which smaller branches branch off. In the case of printed models, the trunk starts at the printer‘s build platform, splits into several branches, and the branches in turn connect to the surfaces of the model. The points of contact between the branches and the model are often referred to as ‚anchor points‘.

12.12 CREATING SUPPORT STRUCTURES WITH PRESET SCRIPTS



Anycubic offers the user three preset scripts to choose from. These can be selected via the drop-down menu in the first line of the window.

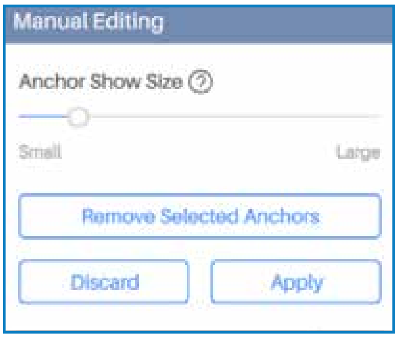
Three variants are available for the preset scripts: „Light“, „Medium“ and „Heavy“. These differ mainly in the number and strength of the support structures. With the „Support Angle“ function, you can define from which angle support structures are to be used, and with „Anchor Distance“ the distance between the support tree trunks. If you click on „Generate Automatic Support“, the support structures are generated automatically. The following three figures show the „Light“, „Medium“ and „Heavy“ variants with a „Support Angle“ of ≤45°.



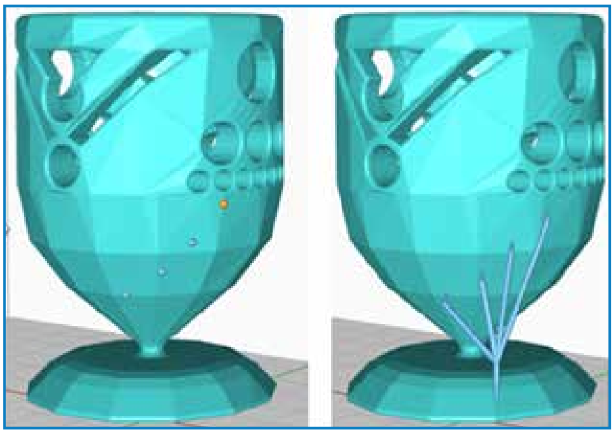
As you can see from the illustrations, this means that the actual model is first printed on its own support structure, with a kind of „perforated plate“ as the base. This has several advantages:

- Printing of the actual model begins only after the transfer layers --> Risk of model warping is minimized
- The perforated base of the support structure makes it easier to remove the model from the build plate.
- Support structures partly start at the absolute Z zero point --> Compensation of possible slight distortion of the model during printing

12.13 MANUAL CREATION OF SUPPORT STRUCTURES



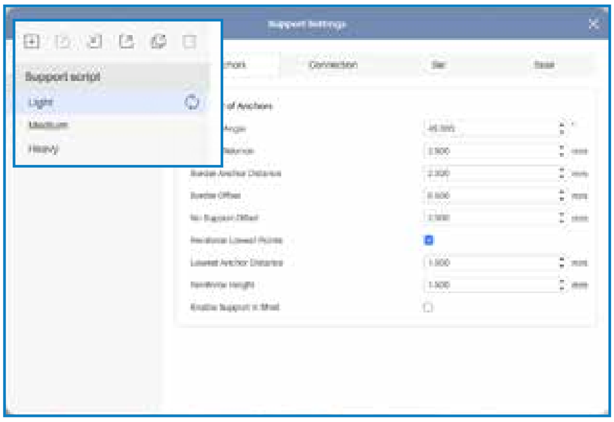
Photon Workshop also offers the possibility to add or remove individual trees of the support structures. Pressing the „Manual Editing" button opens the following window. By clicking with the left mouse button directly on the model, new starting points for the trees can be created or existing ones can be removed. Existing points can also be moved around the model by grabbing them with the left mouse button. In the following illustration, four points are created as an example.



After pressing the „Apply Button" the trees are generated as shown in the picture. For completeness, the applicable shortcuts for using „Manual Editing" are shown again in the following image.

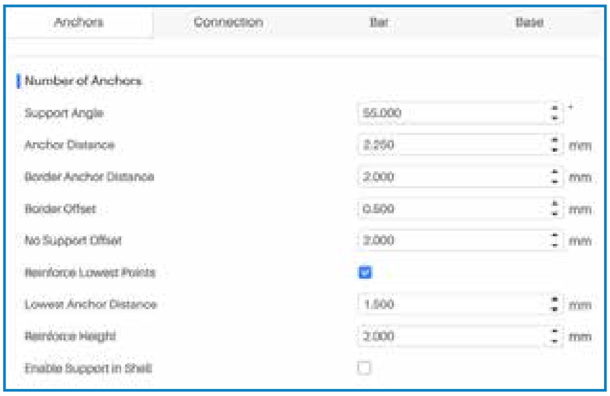
12.14 CREATION AND EDITING OF SCRIPTS

Photon Workshop offers the possibility to adapt the preset scripts for generating support structures individually to the respective model, or to create completely new scripts. Clicking on „More Settings" in the „Support" window opens the following view:

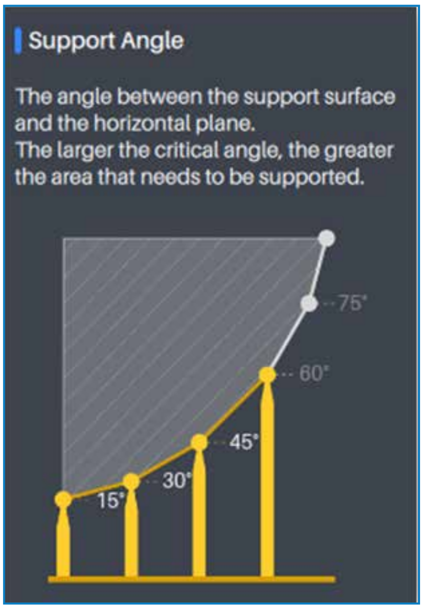


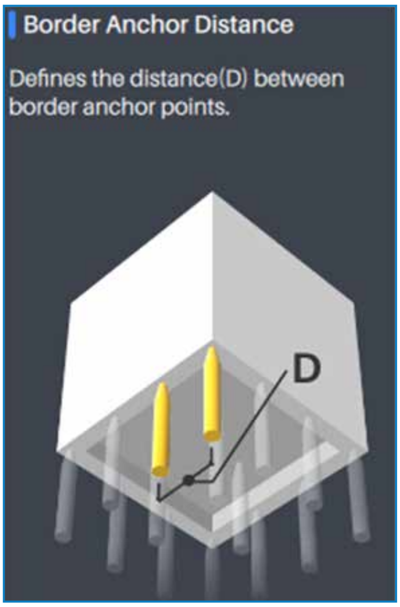
In this menu, new scripts can be created, edited, imported, exported, copied or deleted using the „+" key, and existing scripts can be modified. For advanced users it is recommended to create an individual script for each model. The following sub-chapters explain which settings can be made to generate the support structure.

12.15 SUPPORT SETTINGS - ANCHORS



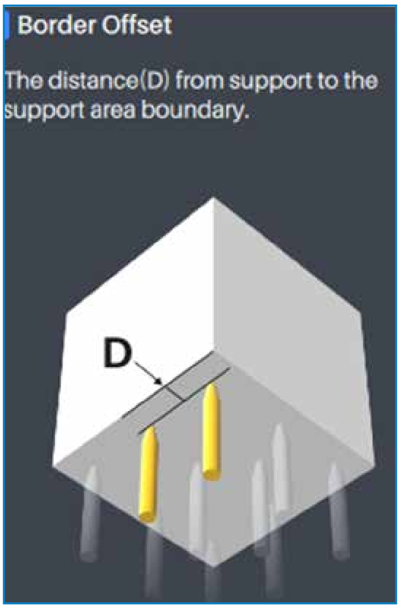
The „Anchors“ menu deals with the settings for the anchor points of the support structure itself.



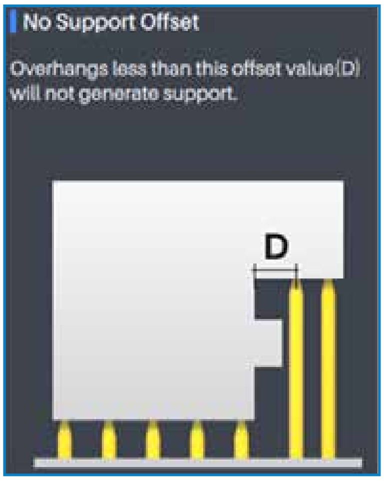


„Anchor Distance“ describes the distance between the inner anchor points. Note: The term „Anchor Distance“ in the previous menu also includes the „Border Anchor Distance“.

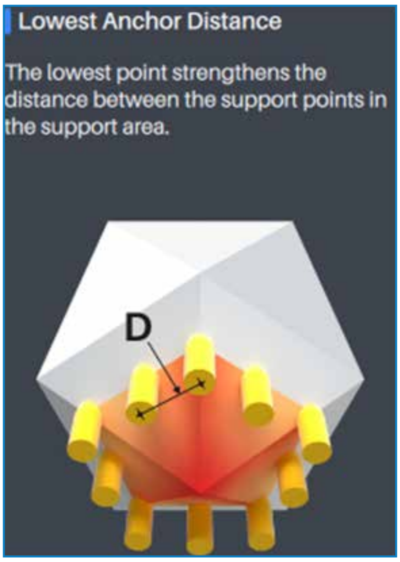
„Border Anchor Distance“ defines the distance between the anchor points of the support structure at the edges of the model. In general, for the „Anchor Distance“, as well as the „Border Anchor Distance“, the distances should be chosen in a way that the model is supported, but the support structures can be easily removed in post-processing, in summary: „As much as necessary, as little as possible.“



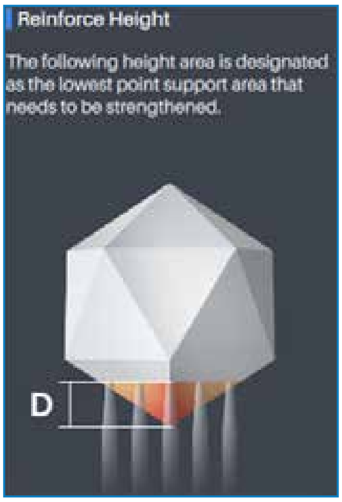
„ Border Offset“ describes the distance from the edges of the model to the anchor points of the support structure. It is advisable to maintain a certain distance from the edges to avoid damage to the edges of the printed model during post-processing, especially when removing the support structures.



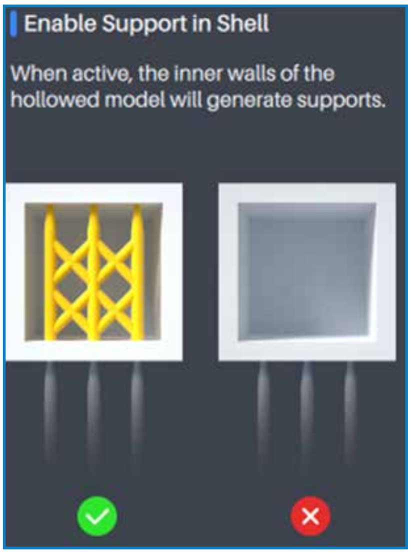
„No Support Offset“ defines the length of overhang that does not need to be supported by a support structure. For example, if the overhangs are short, say ≤2mm, there is usually no need for a support. However, this must be considered on a case-by-case basis, depending on the physical shape of the model and the resin used, and can be more accurately determined by testing.

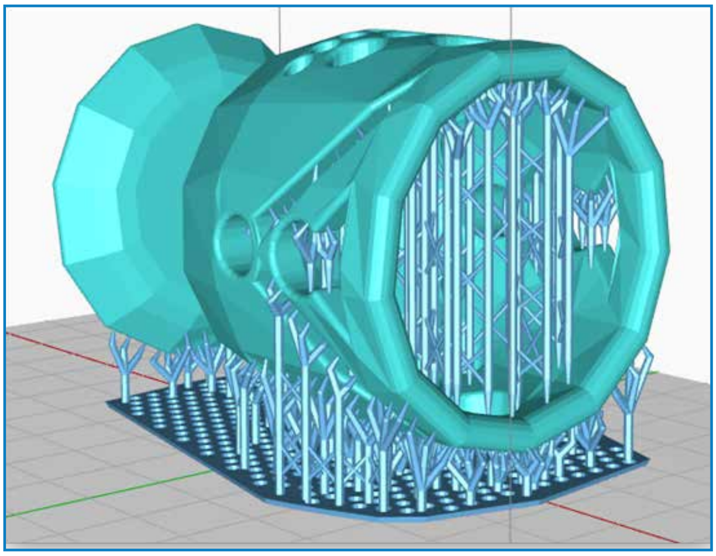


„Lowest Anchor Distance“ defines the distance between the lower ends of the branches. By reducing the distance, the support structure can be tapered.



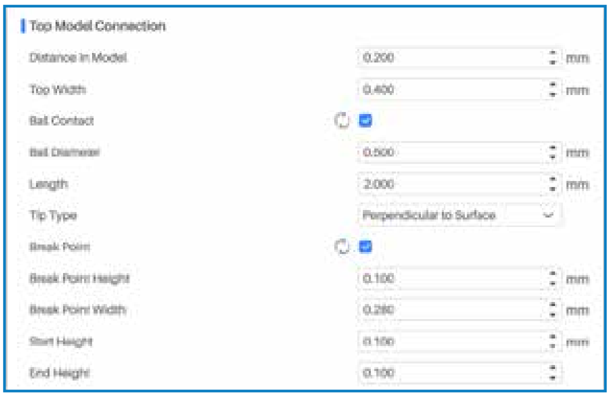
„ Reinforce Height“ describes the length or height at which the model is to be supported by the support structures. The lower the height, the smaller the number of surfaces to be post-processed. Again, the ideal height must be determined by trial and error.





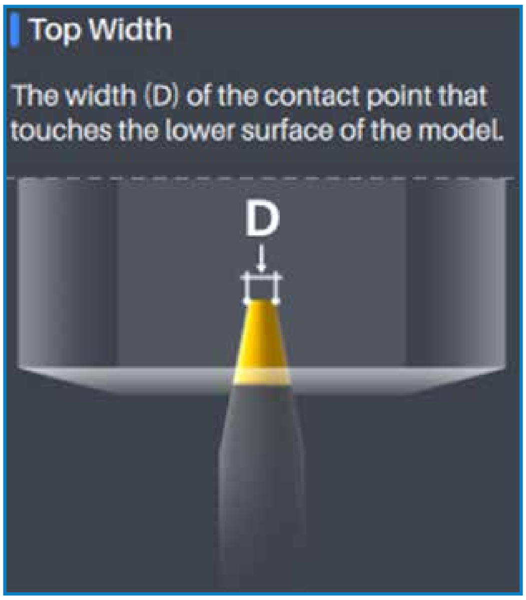
„Enable Support in Shell" is an option that allows the interior of a hollow model to have a support structure. If the egg cup were printed rotated 90° in the Y direction, this option would make the support structure look like shown above.

12.16 SUPPORT SETTINGS - CONNECTIONS

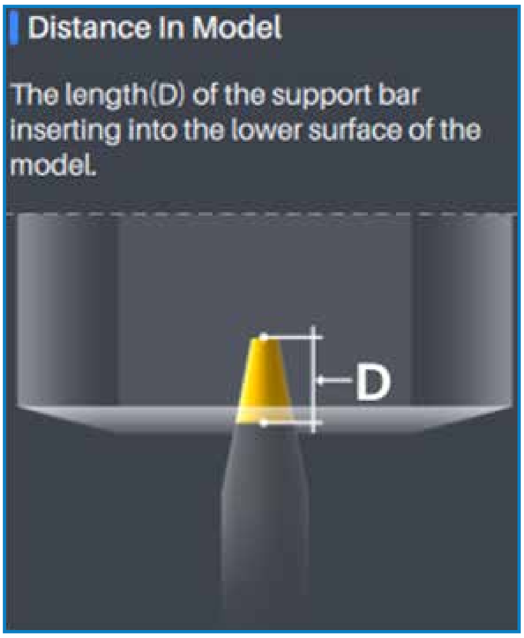


Under the „Connection" tab, the connections between the model and the support structure can be defined in more detail.

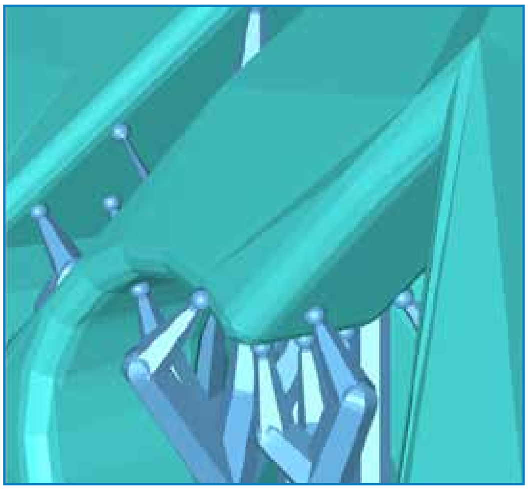
The "Top Model Connection" area defines the top connection points to the model.



„Distance In Model“ defines the length by which the support structure should penetrate the model. This option is particularly important when printing thin-walled models. If the length is too long, the support structures may protrude inside the model.

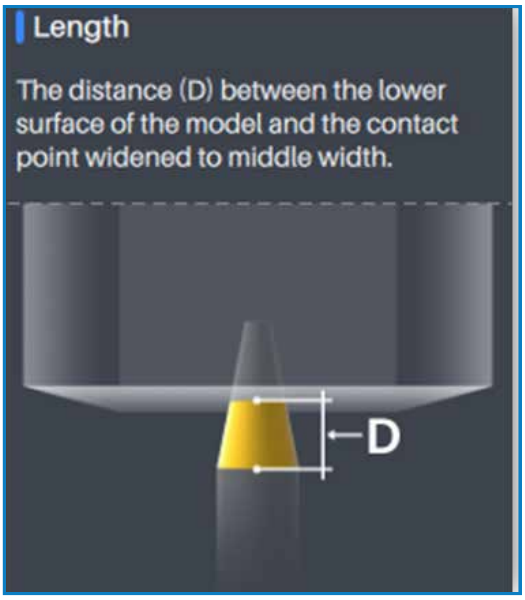


The diameter of the tip of the arm of the support structure can be defined using „Top Width“. The wider the tip, the „tighter“ the connection to the model; the thinner the tip, the easier it is to remove the support structures during post-processing.



Ball Contact

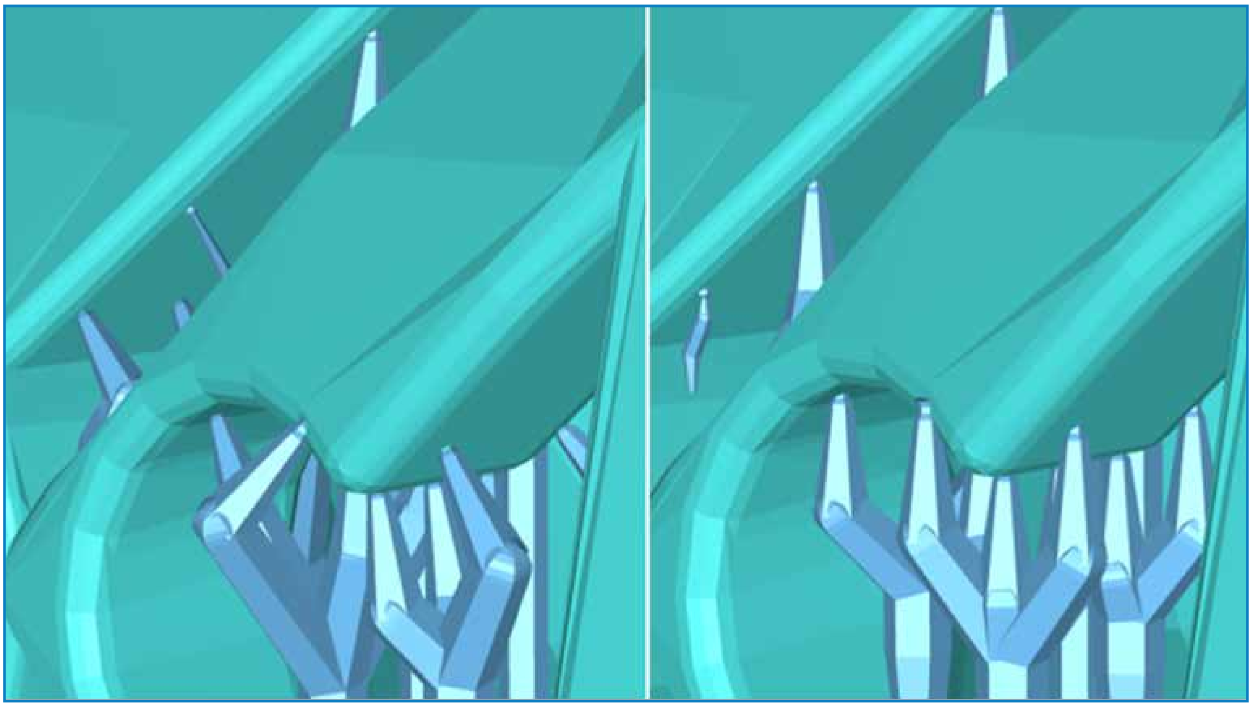
If the „Ball Contact“ option is activated, the tip is spherical. The diameter of the ball can be defined via „Ball Diameter“. In the model, the connections look as follows.



„Length“ defines the distance between the model surface and the initial taper of the support structure branches.

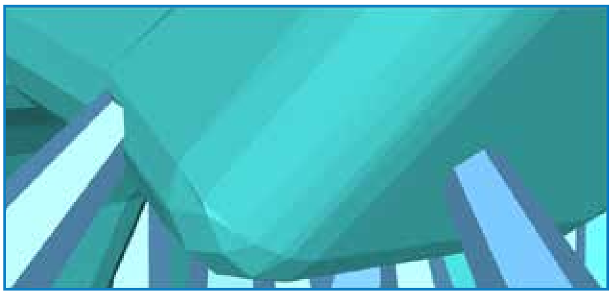
Tip Type

The ‚Tip Type‘ drop down menu allows you to select the direction in which the branches of the support structure should rise from the model. The ‚Perpendicular to Surface‘ option allows the branches to rise perpendicularly to the surface of the model, while the ‚Vertical‘ option allows them to rise perpendicularly to the building platform. The following two figures show the ‚Perpendicular to Surface‘ orientation and the ‚Vertical‘ orientation.



Break Point

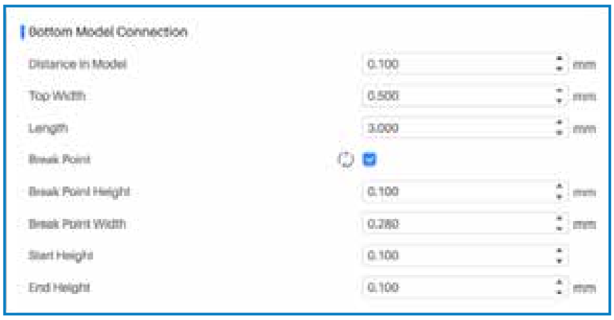
The „Break Point“ function generates nominal break points of the support structures on the surfaces of the model.



The above figure shows the model without predetermined breaking points, in the picture below they are included.

Bottom Model Connection

The bottom model connection points are defined under „Bottom Model Connection“. The setting options correspond to those of the „Top Model Connection“. Only the „Ball Contact“ function is not available here.

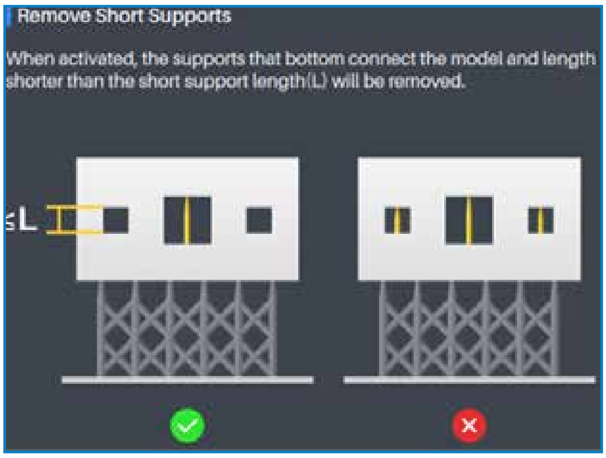


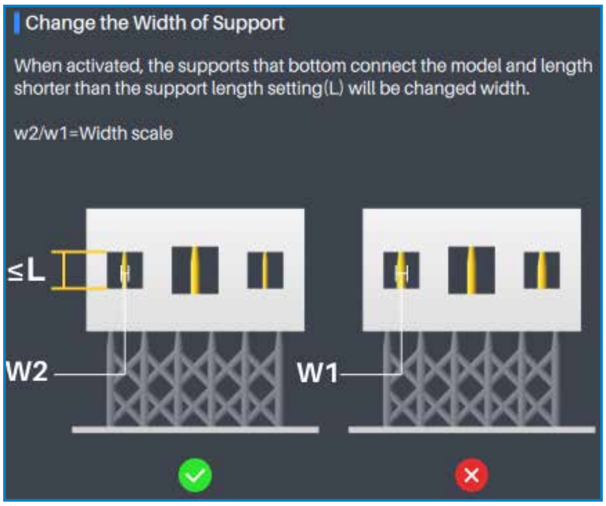
Filter Supports

With the „Filter Supports“ functions, individual support structures can be removed, or automatically adjusted.

Remove Short Supports

The „Remove Short Supports“ function can be used to automatically remove very short support structures, e.g. ≤1 mm. The maximum permissible length of the support structures can be entered directly.



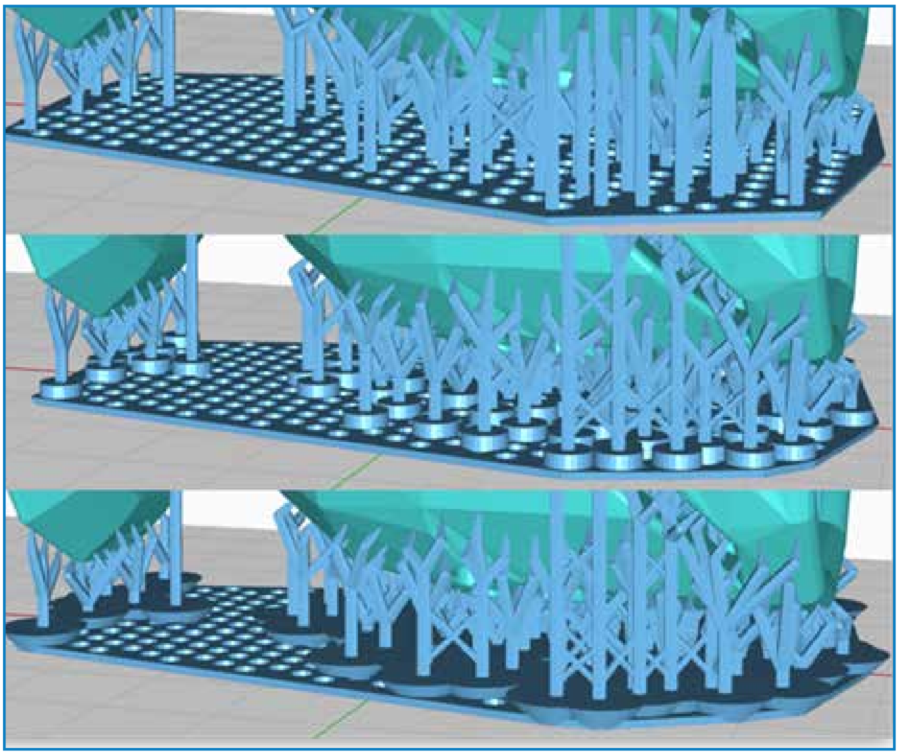


Change the Width of Support

The „Change the Width of Support“ function allows you to change the diameter of support structures that are shorter than a defined length. In doing so, the diameter is scaled smaller by a defined factor. The maximum length as well as the scaling factor can be entered directly in the menu.

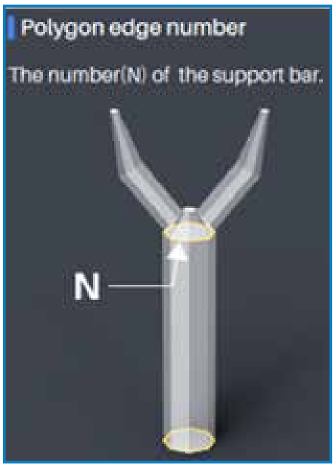
Plateform Connection

Under the „Plateform Connection“ function, you can create your own platforms for the support structures. The height, the diameter and the angle of the platforms can be defined. The following three figures first show the model without own platforms of the support structure, then with small platforms and finally with conical platforms („Slope Angle“ = 45°).

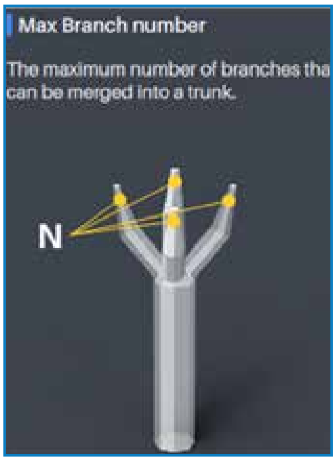


Bar Shape

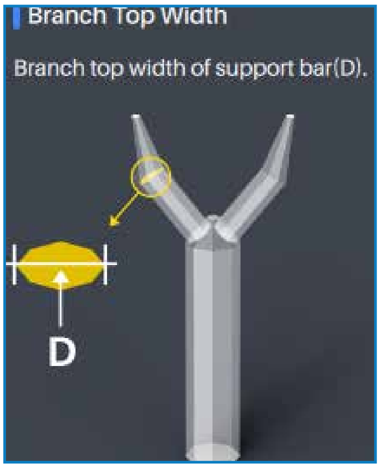
The "Bar Shape" setting is used to specifically address and adjust the geometric shape of individual trees in the supporting structure.



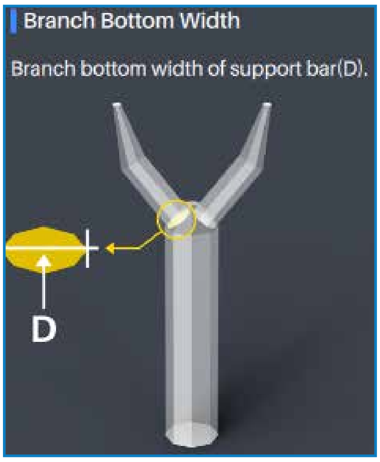
The trees, branches and twigs of the support structure are not „round", but consist of a polygon with n edges in the basic form. The number of corners can be entered directly under „Polygon edge number".



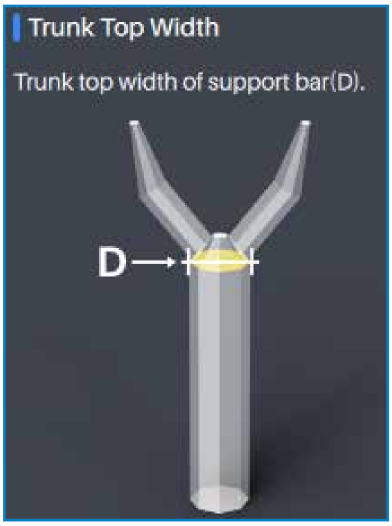
The „Max Branch Number" limits the number of branches on the tree's trunk that „hold" the surface of the model.



The „Branch Top Width" defines the upper diameter of one branch of the tree.

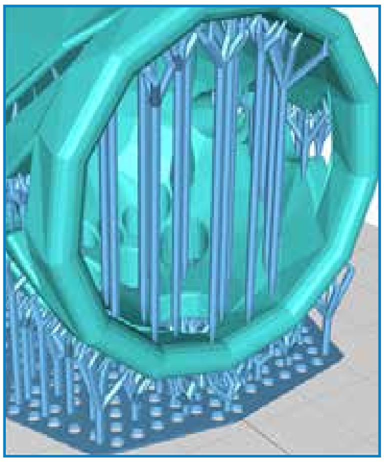


The „Branch Bottom Width" defines the bottom diameter of a branch of the tree. The fact that both diameters can be defined individually means that conical designs of the branches can also be created.



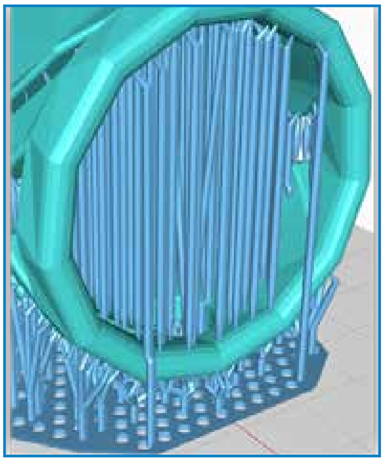
The „Trunk Top Width“ defines the upper diameter of the trunk of a tree.

The „Trunk Bottom Width“ defines the bottom diameter of the trunk of a tree. The fact that both diameters can be defined individually means that tapered versions of the trunk can also be implemented.

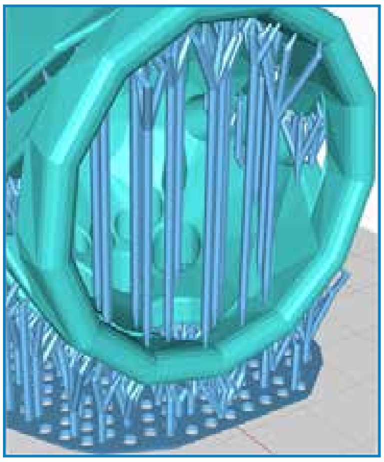


Trunk Height

The „Trunk Height“ can be defined in Photon Workshop using three methods, which can be selected via dropdown. The first option is the „Automatic“ version (see illustration). In this case, the length of the trunk is determined by the programme based on the specified requirements. This method is mainly recommended for relatively inexperienced users.



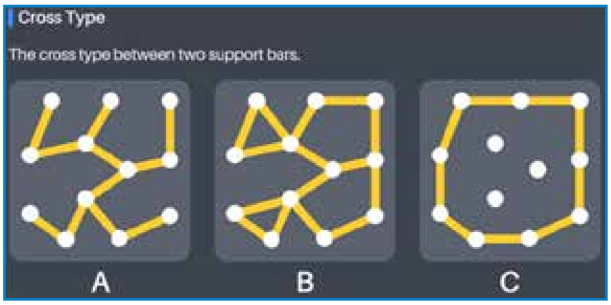
The second option is the “Max Height” method. This defines the maximum height of a trunk. If necessary, additional trees will be added automatically if the length restriction prevents a theoretically successful print.



The third and last option is the „Branch Max Angle“ method. Here, the angle between the branches and the trunk is limited to a certain value. For illustration purposes, an angle of 60° was selected in the following figure.

Bar Cross Connection
When the „Bar Cross Connection“ function is activated, connections can be made between the trees of the support structure.

Cross Type
Cross Type allows you to define the basic shape of the connection between the trees. There are three methods to choose from in the drop-down menu:

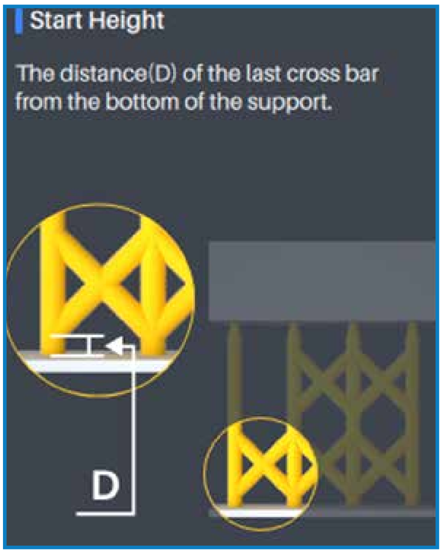


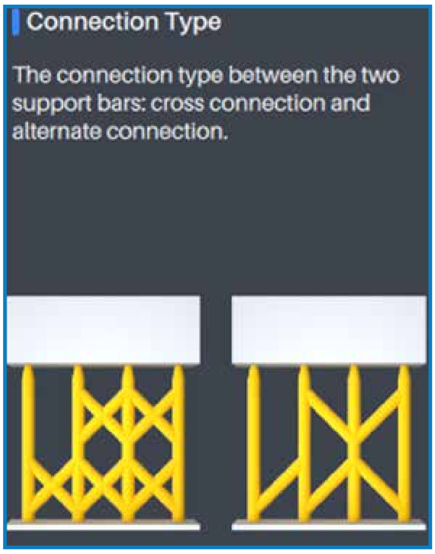
A: „MST Method“ – ensures that all trees are connected with each other, but uses only the minimum number of connections between the trees.

B: „Min2 Method“ – ensures that at least three trees are always connected via two links. This generates accordingly more connections than method A.

C: „Border Method“ – only the trees at the edge of the support structure are connected to each other. Depending on the model to be printed, this can be the method with the least number of connections between the trees.

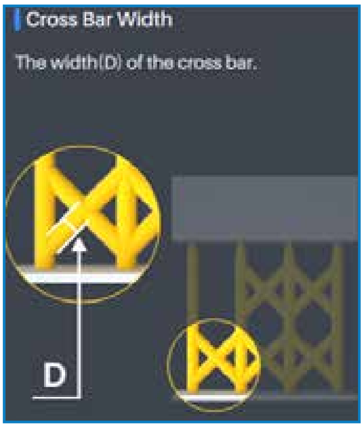
Connection Type

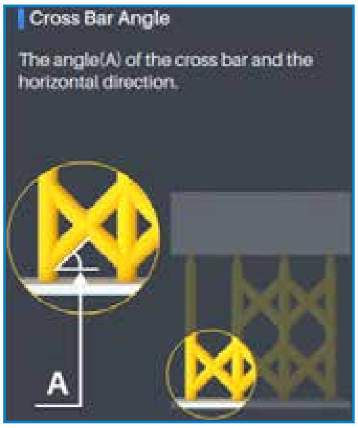


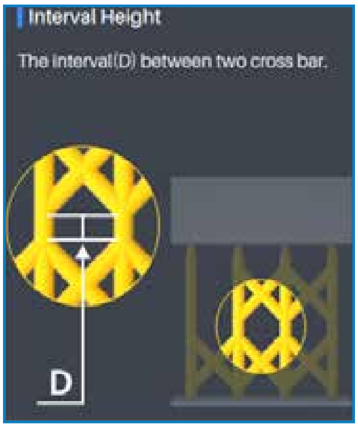


Under ‚Connection Type' you can use the drop down menu to choose between two methods of connecting the trees. The connection shown on the left is the ‚Cross Connection', the connection shown on the right is the ‚Alternate Connection'. „Cross Connection" is the more stable, but also more resin intensive type of connection.

„Start Height" is the distance between the start of the tree and the start of the connection.







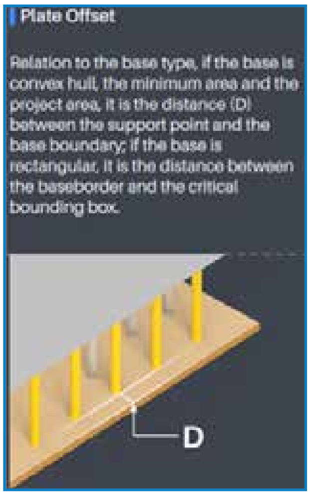
„Cross Bar Width" defines the diameter of the connecting elements.

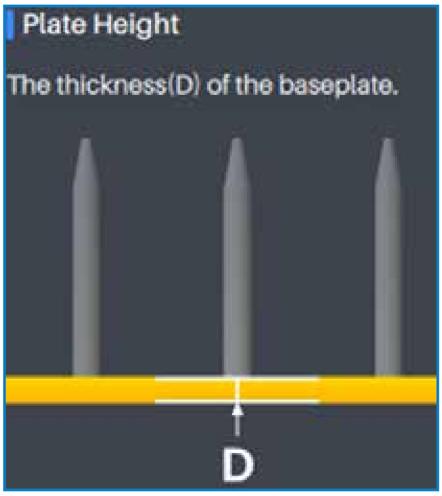
„Cross Bar Angle" defines the angle at which the connecting elements protrude from the trunk.

„Interval Height" defines the distance between two connecting elements.

12.17 SUPPORT SETTINGS - BASE

In the „Base" tab you can create and edit a base that is directly printed onto the build plate of the printer. With this function switched on, the creation of a „base plate" is made possible.





„Plate Height" defines the height oft he base plate.

Slot Angle
„Slot Angle" defines the angle at which the base plate widens towards the top.

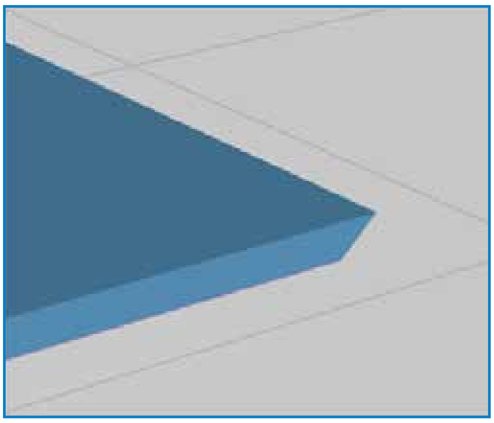
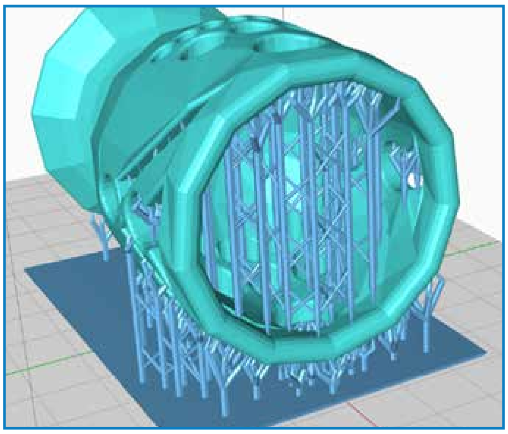


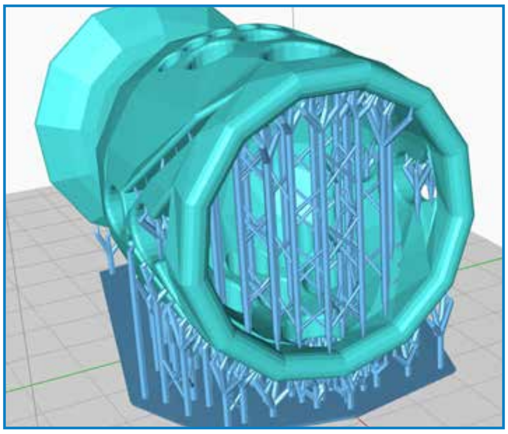
Plate Type

„Plate Type“ defines the geometric base area of the base plate. Photon Workshop offers four different basic shapes via dropdown.

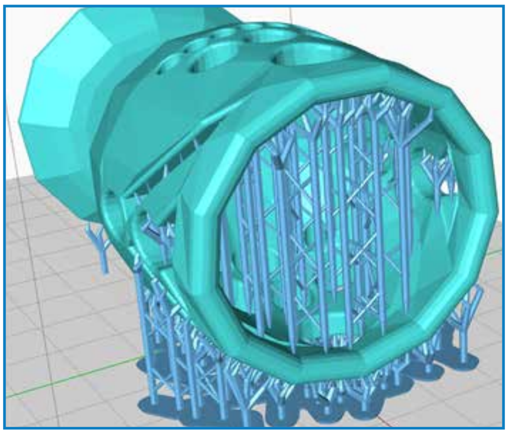
„Rectangle“ defines a simple rectangular base plate. Although this option is the most resin-intensive, it is recommended at the beginning and especially for heavier models.



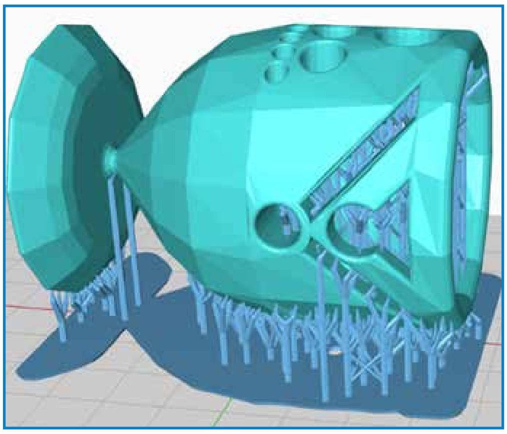
„Convex Hull“ corresponds to a convex shaped projection of the basic shape of the model to be printed.



The „Minimum Area“ option uses only the theoretical minimum area as the base area. Although this is the least resin-intensive option in theory, it can lead to problems with the adhesion of the printed layers to the building platform with heavier models in the advanced printing process.

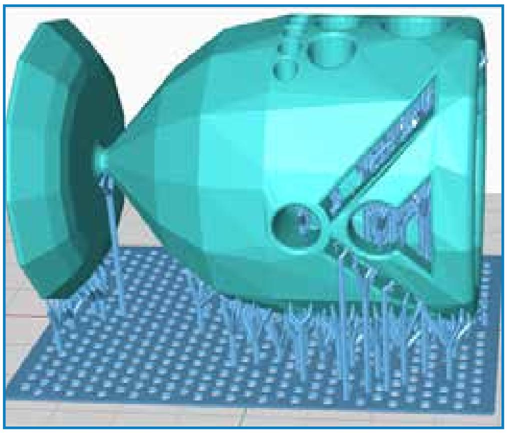


The „Plate Type“ „Project Area“ represents a 1:1 mapping of the projection of the model. Interruptions in the model itself are also mapped (see hole in previous figure).



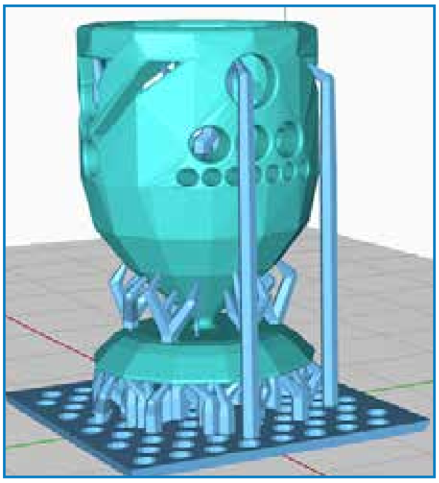
Enable Perforation

The ‚Enable Perforation‘ function allows additional holes to be made in the base plate, regardless of the ‚Plate Type‘. This saves resin and makes it easier to remove the printed model from the building platform. The diameter of the holes can be defined by the „Hole Radius“. The number of holes is determined by the „Hole Interval“ which defines the distance between the holes. As these holes are not „round“, but are polygonal, the number of corners can be defined by „Hole Edge Number“. As with the trees of the support structure, the more corners defined, the „rounder“ the hole. The following is an example of a baseplate with holes.

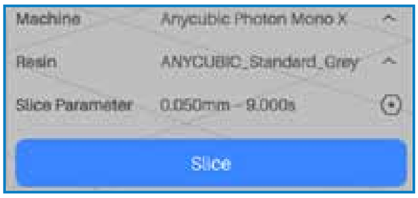


12.18 SLICING OF THE MODEL

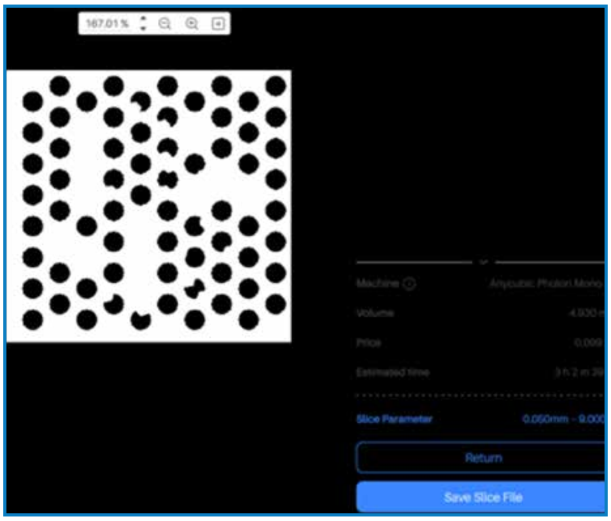
Once all the parameters of the model have been set, the model should be displayed as an example in Photon Workshop as follows.



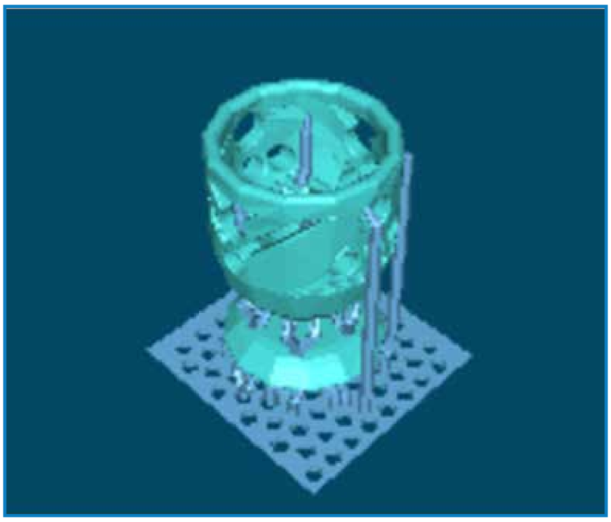
In the next step, the slice process can be started.



The slicing process is started by pressing the ‚Slice‘ button to the right of the machine and resin parameters. The duration of the slicing process depends on the characteristics of the model and the processing power of the PC. It usually takes only a few seconds on older machines and large models.

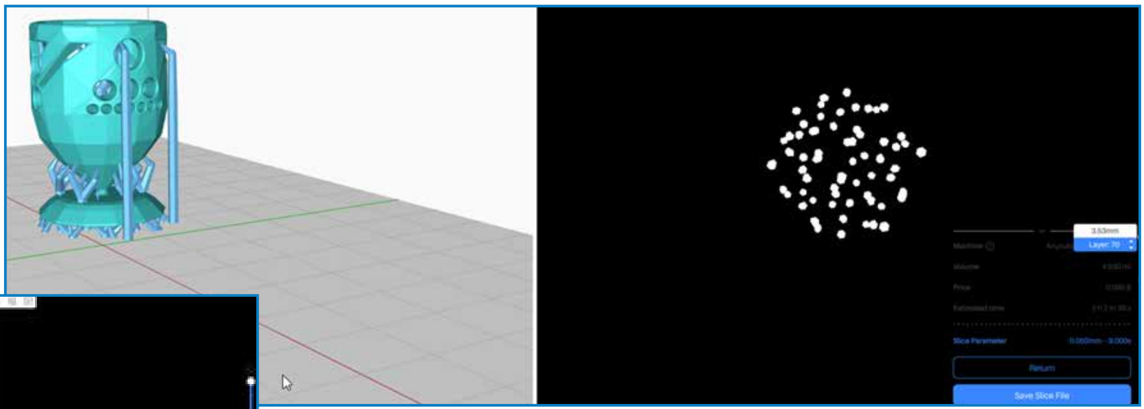


When the slice process is complete, the view shown is displayed. At the bottom right, the estimated resin consumption is shown, as well as the estimated material costs and the estimated printing time. On the left side of the image, the first layer of the print is shown from the top view. Using the slider on the right side, the different layers of the print as well as the respective height of the model can be displayed. In addition, the buttons at the top of the image can be used to switch to other views or to activate auxiliary functions



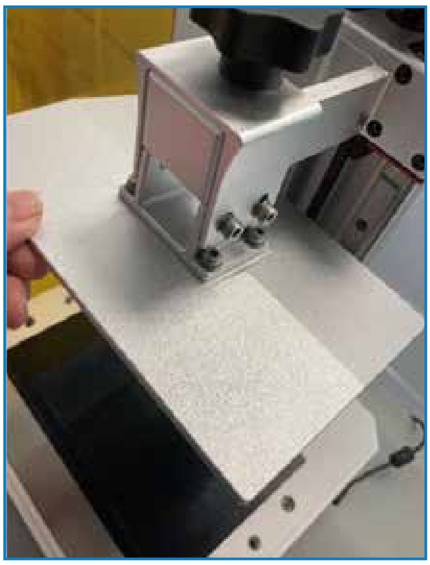
The „Preview Image“ function shows the print model again.

The function „Check Islands“ is a useful tool for the final check of the model before printing. Here the layer areas shown on the right are coloured according to a traffic light system (see below).



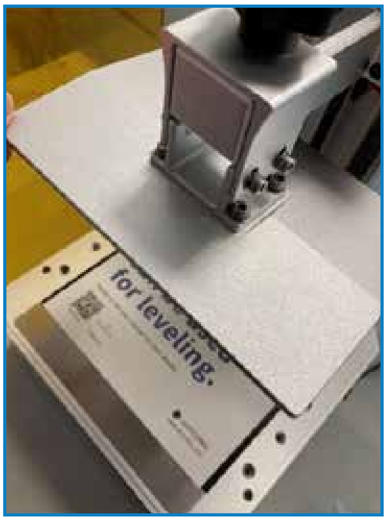
The system checks that all areas of the layer are generally printable. If the colour is green, the system has no problems printing. If it is yellow, it may not be possible to print cleanly. The edges of the model are often coloured yellow. This is usually due to the edge of the model being on the pixel boundaries of the UV display and therefore complete curing cannot be guaranteed. If areas are coloured red, they are not considered printable by the system. At this point it is recommended to check all layers of the model using the slider. If there are red areas, you can use the „Return" button to return to the settings and make the red areas printable, for example by adding more support structure elements. If no red areas are visible in the layers, the ‚Save Slice File' button can be used to save the slice as a print file to the PC hard drive or directly to a USB stick for later printing. The file is saved directly in .pwmx print format.

12.19 PHOTON MONOX CONDITIONS



The levelling of the Z-zero is essential to carry out the print. If the build platform is too far from the bottom of the resin basin when printing starts, the first printed layer cannot adhere to the build platform and remains on the FEP film. If the build platform runs against the bottom of the resin basin, in the best case the emergency stop of the Z-drive will end the run, in the worst case the build platform will run against the UV screen and damage it.

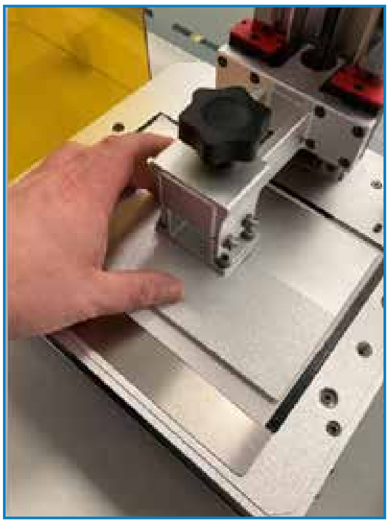
In the first step, it is therefore necessary to loosen the four horizontally aligned hexagon socket screws on the build platform to such an extent that it can be freely and easily swivelled and pushed upwards. At the same time, however, it must also be ensured that the build platform itself is firmly fixed to the Z-arm of the printer. To do this, tighten the vertical knurled screw hand-tight.



The resin basin is then removed and the levelling paper is placed over the UV screen.

Now you can switch on the Mono X and position the spindle on Z0. To do this, first open the „Tools“ category in the start menu and then the „Move Z“ menu.

Then press the „Home“ button (house symbol).



The spindle then moves in the Z direction to its absolute zero position. The building platform is pushed upwards when it comes into contact with the levelling paper because it has been loosened in advance. By gently holding down the building platform with the fingers, the four horizontal hexagon socket screws can now be tightened again. After tightening, the button „Z = 0“ must be pressed.

When the new zero point is confirmed, the build platform is raised.

12.20 SOFTWARE UPDATES

The Anycubic Photon Workshop and the Anycubic Photon Mono X should always be at the same software level, which means that the slicer software and the printer software must be compatible with each other from a versioning point of view. If this is not the case, printing cannot be carried out or the models may not be printed completely.
Photon Workshop notifies users when a new software version is available. When updating the Workshop, it is therefore worthwhile to take a look at the manufacturer‘s website to see if a firmware update is also available for the Mono X printer.

Firmware updates for Anycubic printers and systems can be found at https://www.anycubic.com/pages/firmware-software or under the following QR code.

All you have to do now is select the correct printer type. At the time of writing this manual, you are then forwarded to a Google-Drive-Folder. The .zip file shown there must now be downloaded as the first step. The .zip file contains four files and one .txt file.

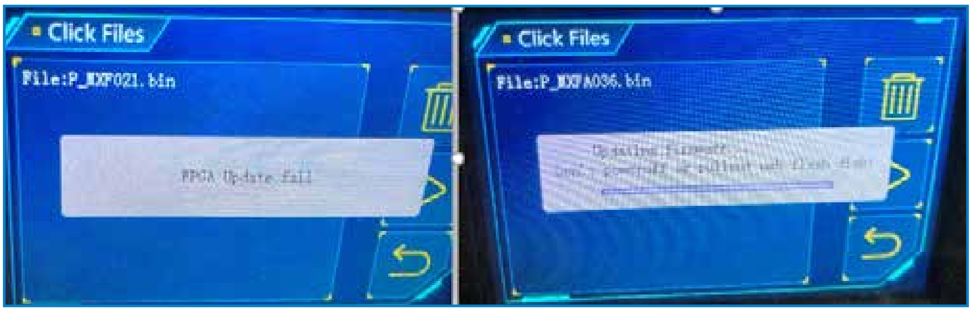
- P_MXF021.bin
- P_MXFA036.bin
- P_MXV5_4_0812(1).bin
- Read me

The .bin files must be loaded onto a USB stick and inserted into the USB port of the printer, the .txt file is an installation guide for the firmware update.

To install the firmware update, the .bin files must now be printed in the correct order in the printer.

You have to start with the „P_MXV5_4_0812(1)“ file.

Next is the file „P_MXF021"



If the installation is not successful or is stopped by the printer, use the file „P_MXFA036". The Mono X should then restart itself and complete the installation of the firmware update.

Unit 13

Printing Process

In the following subchapters, the actual printing of a model is explained in more detail using an example.

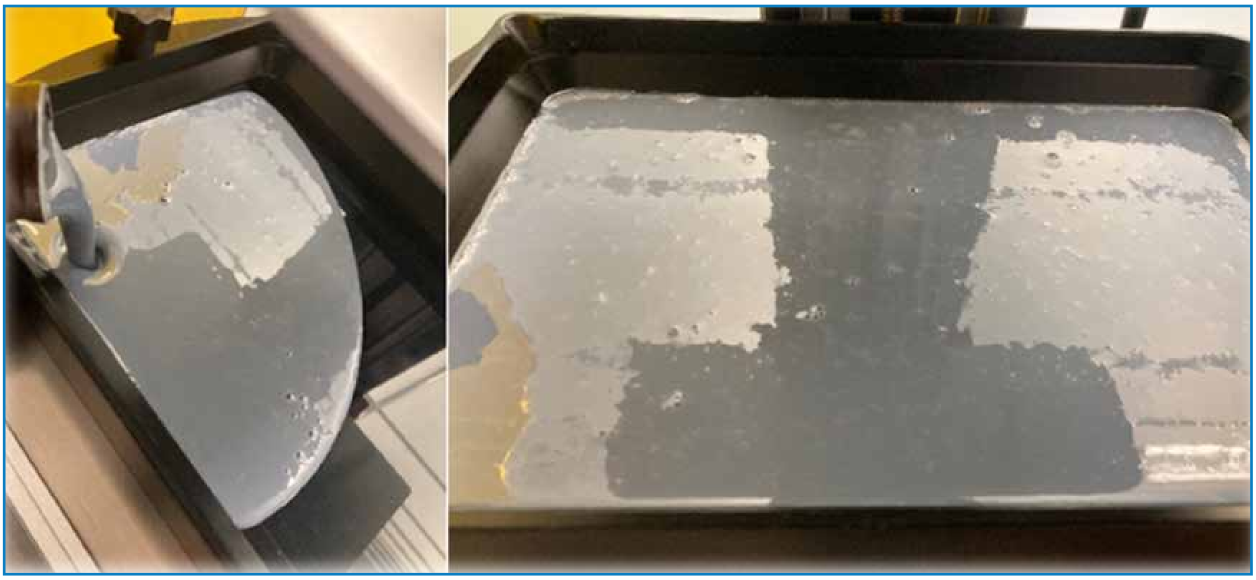
Information

First of all, it should be mentioned that the **use of the printer and the handling of resin and solvents (especially isopropanol) is only recommended when wearing protective gloves (e.g. disposable gloves) and protective goggles in well-ventilated rooms** due to the health risks associated with direct skin contact.

13.1 PREPARATION PHOTON MONO X AND START OF THE PRINTING PROCESS

First, the printer itself must be prepared for printing. To do this, the resin basin must first be inserted and fixed in place with the two knurled screws. In the next step, the build platform is pushed onto the Z-arm of the printer and fixed in place using the large knurled screw.

In the next step, the used resin is shaken up well. Make sure that the resin bottle is well closed. After shaking, the resin can be poured into the resin basin. Make sure that the marked maximum line at the edge of the basin is not exceeded, otherwise the resin will overflow when the build platform is immersed and can damage the printer, especially the UV screen.

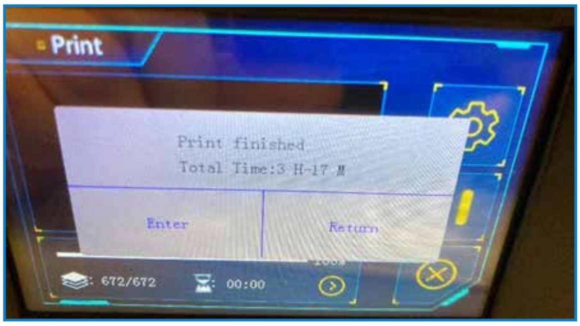


Shaking the resin in the bottle has caused air bubbles to form in the resin. It is strongly recommended to wait until the air bubbles have disappeared before printing.

Once the resin has settled in the basin, the protective cover can be placed on the printer and the printing process can begin. First insert the USB stick with the corresponding print files into the USB port of the printer. Then select the „Print" button on the start page of the menu. In the following menu, the print files stored on the USB stick are listed.

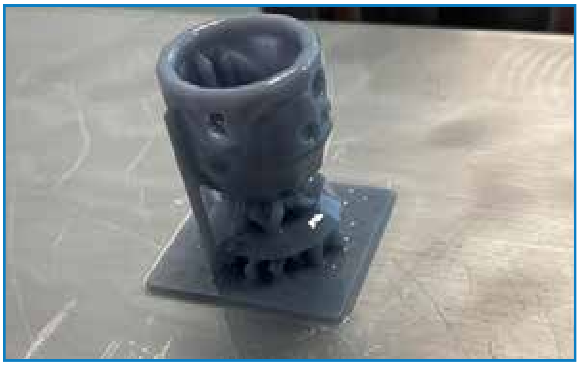
On the start page of the menu, select the „Print“ button. In the following menu, all print files stored on the USB stick are listed. After you have selected the desired print file, a preview of the print process is displayed. If everything is correctly selected, you can start the printing process by pressing the „Start“ button. The print screen shows you the current layer to be printed, the number already printed and the expected remaining print time. Using the settings menu in the upper right-hand corner, you can change various print parameters again afterwards.

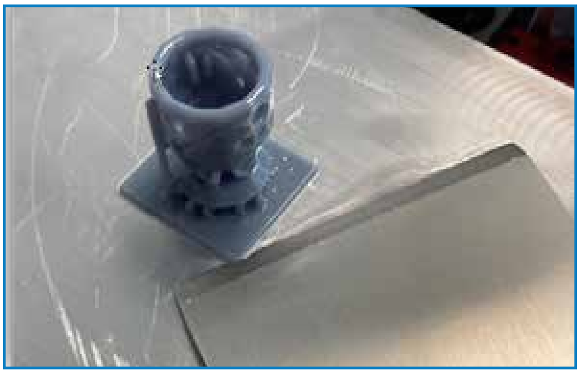
13.2 HOW TO CLEAN ANYCUBIC MONO X AFTER PRINTING

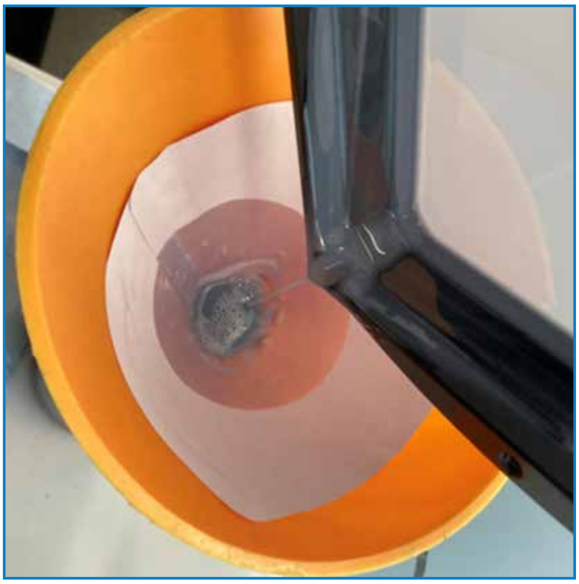


When printing is finished, the Z-arm of the printer returns to its original position. After confirming the print on the touch screen, the protective cover can be removed.

The printed model is removed from the build platform using a spatula. It is a good idea to turn the build platform upside down, i.e. remove the build platform from the Z-arm of the printer and then place it back on the Z-arm with the printed model on top.







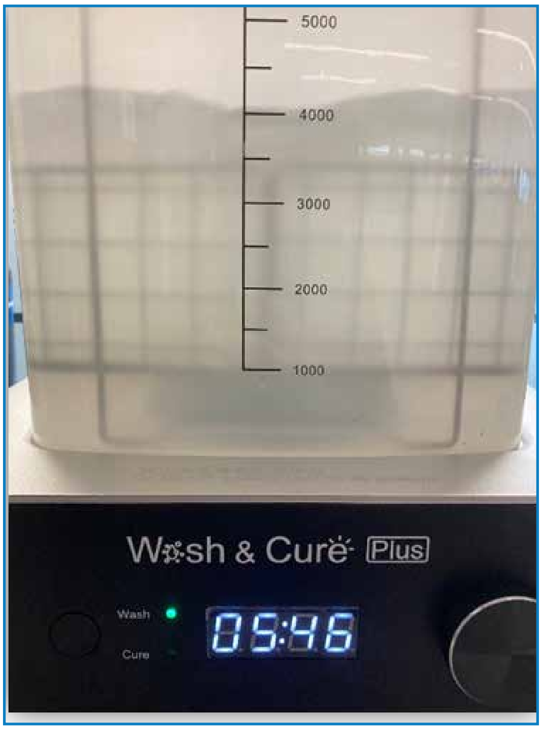
After removing the printed model, the building platform can be removed and put aside. In the next step, the resin remaining in the resin basin is filled back into the resin bottle. To do this, remove the knurled screws and lift the resin basin from the printer. The resin is refilled via a funnel with an inserted ink sieve to prevent hardened resin residues in the resin basin from flowing back into the resin bottle. To facilitate pouring out the resin, a pouring nose is attached to the resin basin. Finally, the building platform and the resin basin are cleaned with isopropanol. It is recommended at this point to check the cleanliness of the printer and especially the operating surfaces with a UV light after cleaning in order to avoid direct, unnoticed contact with the resin used.

13.3 CLEANING, CURING AND POST-PROCESSING OF THE PRINT

Information

The following subsections explain how to clean the printed model and how to finish it. As with the previous point, it is recommended to work with protective gloves and goggles in well-ventilated rooms. A protective mask (e.g. FFP2 or FFP3) is also recommended for any grinding work for post-processing.

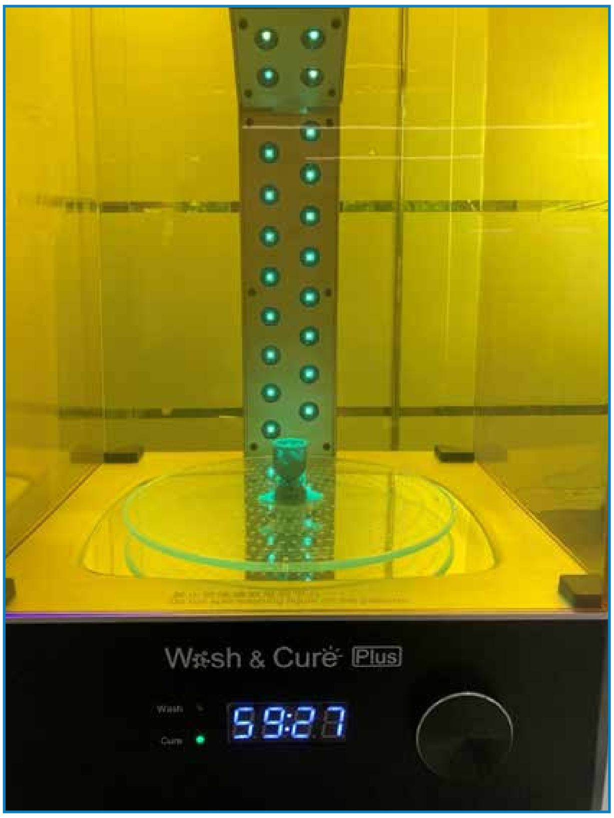
After removing the printed model from the building platform, place it in the basket of the ‚wash' container and place the basket in the isopropanol bath. The isopropanol bath should be filled with isopropanol to about 5 cm above the model. Close the isopropanol bath. Pay particular attention to the tightness and correct fit of the lid. When the isopropanol bath is closed, place it on the Wash & Cure unit. Ensure that the drive shaft-hub connection is correctly seated.



Set the „Wash & Cure" unit to „Wash" and set the wash time by turning the knob. The wash time to be set depends on the size and complexity of the printed model. However, a minimum wash time of 6 minutes is recommended. This recommendation is due to the fact that the washer reverses the direction of rotation, i.e. the direction of flow of the isopropanol, every two minutes. This ensures good cleaning even for smaller, complex models.

After the machine washing process has been completed, the model can be removed. It is recommended to rinse the model briefly with fresh isopropanol.

13.4 CURING OF THE PRINTED MODEL



After the printed model has been cleaned, the „Wash & Cure“ unit is converted to „Cure“ operation. To do this, the isopropanol bath is lifted down, the mirror foil is inserted and the turntable is put on. Here, it is also important to ensure that the shaft-hub connection is correctly seated.

The printed model is placed on the turntable, the protective cover is pulled over and the „Wash & Cure“ unit is set to „Cure“. The curing time can be set the same as the washing time on the unit. The curing time also depends on the size and complexity, but especially on the wall thickness of the printed model. A minimum curing time of 45 minutes is recommended here. After the curing process is finished, the protective cover can be removed and the model can be taken out.

If white or generally lighter areas remain on the model, these can be carefully washed off afterwards with a brush soaked in acetone. These lighter areas are the result of resin residues that were not completely washed off and were finally "hardened" to the printed model by the curing process.

Cleaning of the used Isopropanol

After several machine washes of models, the isopropanol used in the isopropanol bath becomes heavily contaminated with resin residues adhering to the models. This encourages the formation of the aforementioned light spots on the cured models. Therefore, it is recommended to clean the isopropanol bath regularly.

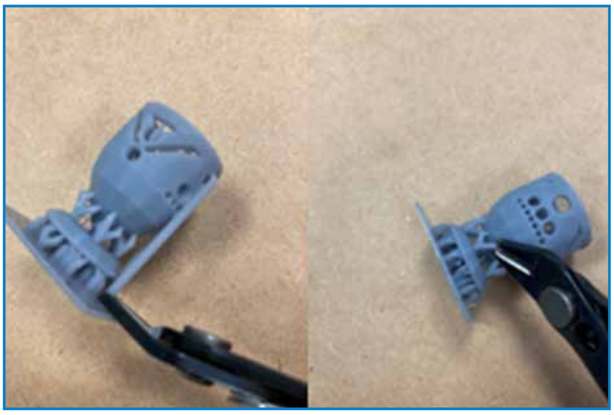
The cleaning procedure is as follows: Pour the contents of the isopropanol bath into a suitable clear container. The container should then be left in a well-lit room for a few days. As the resins have a higher density than the isopropanol, the resins sink to the bottom of the container where they are cured by natural UV radiation. After a few days, a clear separation of the resin residues and the isopropanol can be seen. Meanwhile, the isopropanol bath itself is cleaned with isopropanol and cloths.

The now purified isopronal can be poured back into the isopropanol bath through a colour sieve.

The cured resin residues remain in the container after pouring.

13.5 REMOVAL OF THE SUPPORT STRUCTURE

After curing, the support structures on the printed model can be removed. An electronic side cutter, a wallpaper knife or scalpel with a stiff blade and sandpaper in different grit sizes are recommended for this purpose.

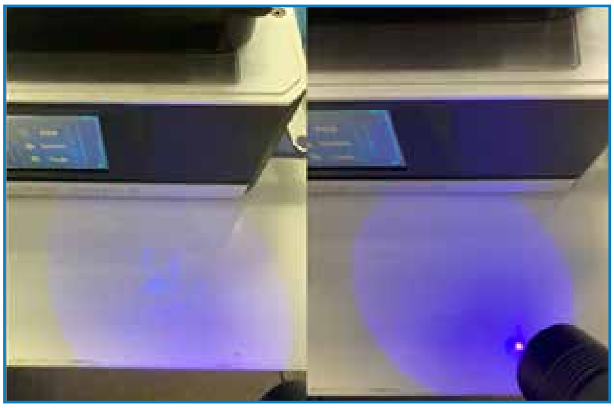


In the first step, the rough support structures are removed with the electronic side cutter.

Then carefully cut off the stumps of the support structure with the wallpaper knife.

If necessary, use sandpaper to carefully sand the connection points of the previously cut stumps.

13.6 CLEANING THE WORKPLACE



After completing the work, the workplace should be cleaned with isopropanol or a suitable industrial cleaner, and checked for spots with a UV torch.

Despite wearing protective gloves, hands must be washed thoroughly and then checked with a UV torch.

TRAINME

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