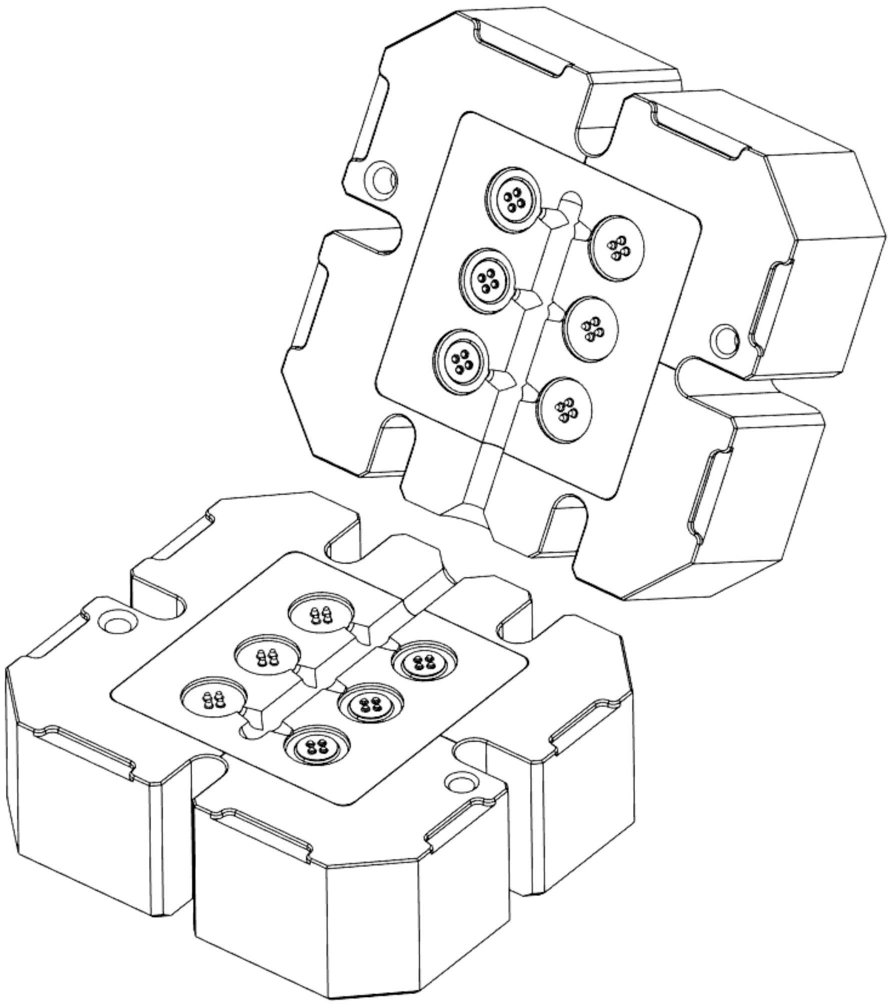
plasticpreneur®

UNIVERSAL MOULD DESIGN GUIDE

FOR 3D PRINTED AND ALUMINIUM MOULD INLAYS

70x70x50 mm

120x100x40 mm



Version 1

Introduction

With the plasticpreneur Injection Moulding Machine and the Universal Moulds, the gap between 3D printing and industrial injection moulding is effectively bridged.

Injection moulding is ideal for producing components in large quantities with consistent quality. However, the high initial cost of producing injection moulds is a major disadvantage, particularly for prototyping and small batch production, making the technology expensive for these applications.

3D printing, on the other hand, is perfect for fast and cost-effective prototyping, as it does not require high setup costs. However, the quality and strength of 3D-printed parts are lower compared to injection moulded parts. Additionally, 3D printing is a relatively slow manufacturing process, making it uneconomical for series production beyond a certain quantity.

This is where the plasticpreneur Injection Moulding Machine comes into play. It offers a much more affordable alternative to traditional injection moulding machines, as the initial mould costs are significantly lower and the production cycle is considerably shorter compared to 3D printing. This effectively closes the gap between 3D printing and industrial injection moulding.

By combining Universal Moulds with 3D-printed inlays, these two technologies can be effectively integrated, making the solution ideal for cost-effective prototyping and small series production. The aluminium frame of the Universal Mould not only ensures even distribution of injection pressure but also acts as a buffer between the hot nozzle and the 3D-printed inlay, thereby maximizing the inlay's service life.

The plasticpreneur Universal Mould is available in two sizes:

- 70x70x50 mm
- 120x100x40 mm

Depending on the size of the part to be produced, the appropriate mould size should be selected. Choosing the right size helps to reduce both the material consumption and the production time required for the inlay.

Larger Universal Moulds are theoretically possible. However, larger 3D-printed inlays are more prone to warping, which can lead to significant post-processing efforts and flashing. For this reason, the production of larger Universal Moulds has currently been discontinued.

Mould Inlay Manufacturing Methods

There are three main methods suitable for manufacturing mould inlays:

SLA 3D Printing (Stereolithography)

In this process, a liquid photopolymer resin is selectively cured using a laser, UV light and heat. The combination of high detail resolution and good heat resistance makes SLA printing especially suitable for producing high-quality mould inlays.

Advantages:

- Very high surface quality
- High dimensional accuracy and stability
- Suitable for complex geometries
- Relatively fast production
- Good price-performance ratio (€€)

Disadvantages:

- Resin materials are comparatively expensive
- Not recyclable

Recommended Materials:

- Formlabs Rigid 10K
- Phrozen Functional TR300

FDM 3D Printing (Fused Deposition Modeling)

In this method, a thermoplastic filament is melted and deposited layer by layer to form a solid part. FDM printing is particularly cost-effective, but it offers lower precision and mechanical strength compared to SLA.

Advantages:

- Very low cost (€)
- Fast manufacturing
- Suitable for moderately complex geometries

Disadvantages:

- Visible layer lines
- Lower dimensional accuracy
- Reduced lifespan compared to SLA inlays

Recommended Materials:

- Polycarbonate
- Nylon (note: limited durability)

CNC-Machined Aluminium Inlays

Similar to traditional injection moulding, inlays can also be manufactured from aluminium using CNC milling. These inlays offer excellent surface quality and high durability while being more affordable than a fully CNC-machined injection mould.

Advantages:

- Very high dimensional accuracy
- Excellent surface quality
- High durability

Disadvantages:

- Higher production costs (€€€)
- Longer manufacturing times

Recommended Material:

- Aluminium 7075

Universal Mould Design Guide

Step 1: Product Design

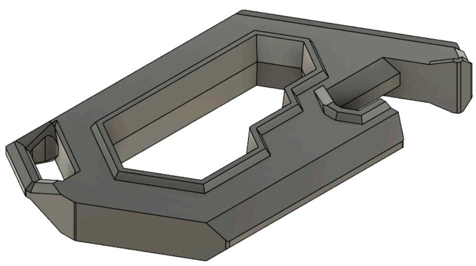
Create the 3D model of the product you intend to manufacture using the injection moulding process. Make sure the dimensions fit one of the two Universal Mould variants:

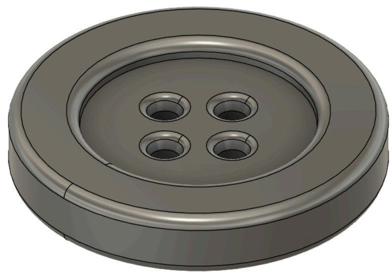
- 70 x 70 x 50 mm
- 120 x 100 x 40 mm

Pay particular attention to the following design guidelines for injection moulding:

- Draft angles of at least 5–10° to ensure easy demoulding (especially important for 3D-printed inlays with visible layer lines).
- No undercuts, to avoid the need for technically complex demoulding mechanisms.
- Sliders and inserts are possible but may require additional post-processing due to manufacturing tolerances in 3D printing, to ensure smooth functionality.

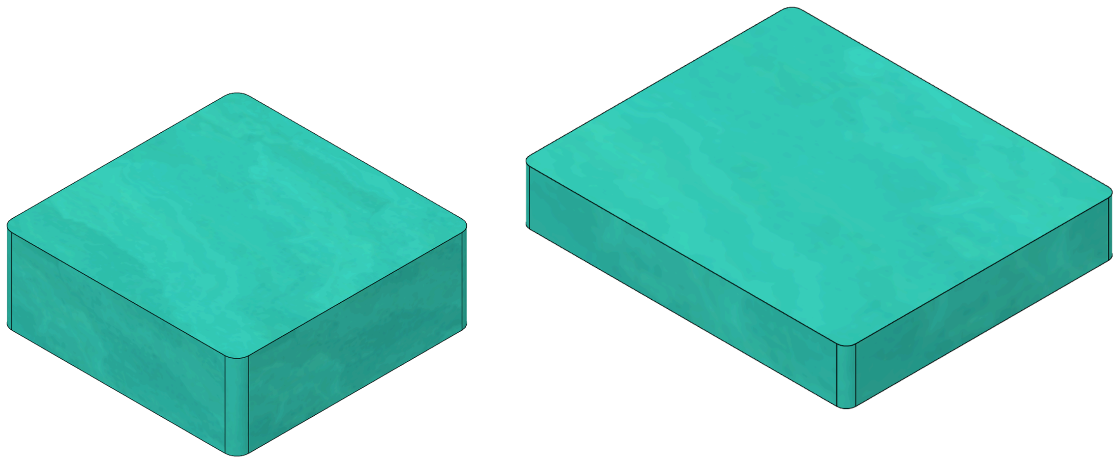
In this guide, we use a multitool and 15 mm buttons as example products.





Step 2: Preparing the Inlay Base File

Use one of the provided **inlay base files** as a starting point. Depending on the size and shape of the product, it is also possible to combine multiple cavities into a single inlay



Step 3: Choosing the Injection Point

Select a suitable injection point based on:

- The geometry of the product
- The number and arrangement of the cavities

The Universal Moulds offer two injection options:

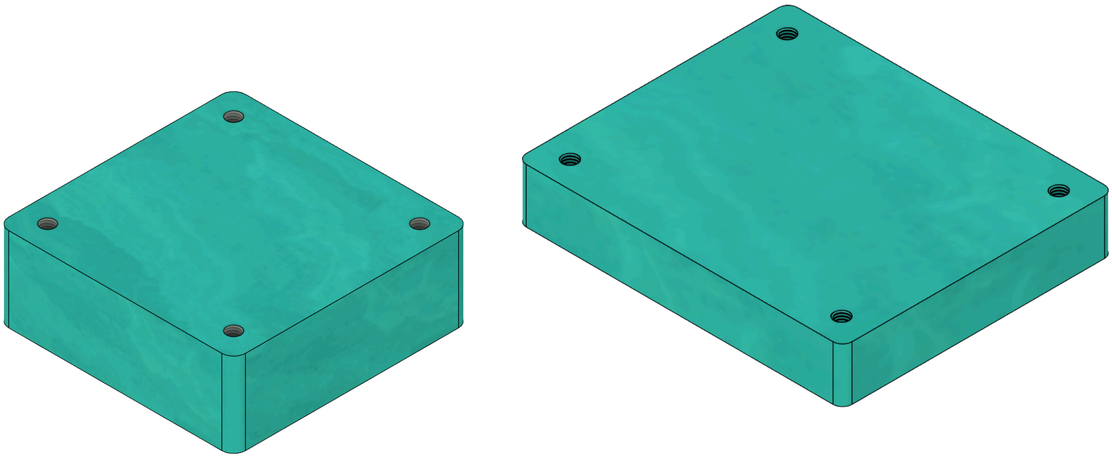
- **Side injection at the parting line**: Simplifies demoulding.
- **Direct injection through the aluminium frame**: Slightly more complex, but required for certain geometries.

The inlays are mounted using four **M5 screws**, which are screwed directly into the aluminium frame.
The threaded holes are already integrated into the base files.

Screw hole spacing:

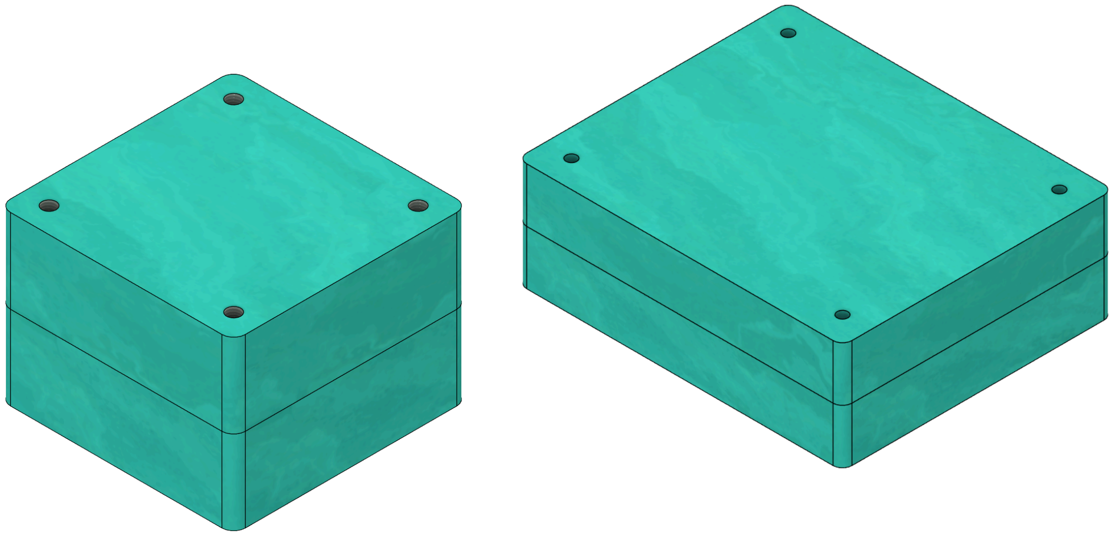
- 55 x 55 mm for the 70x70x50 mm variant
- 100 x 80 mm for the 120x100x40 mm variant

If a cavity overlaps with a screw position, the corresponding screw hole can be removed.



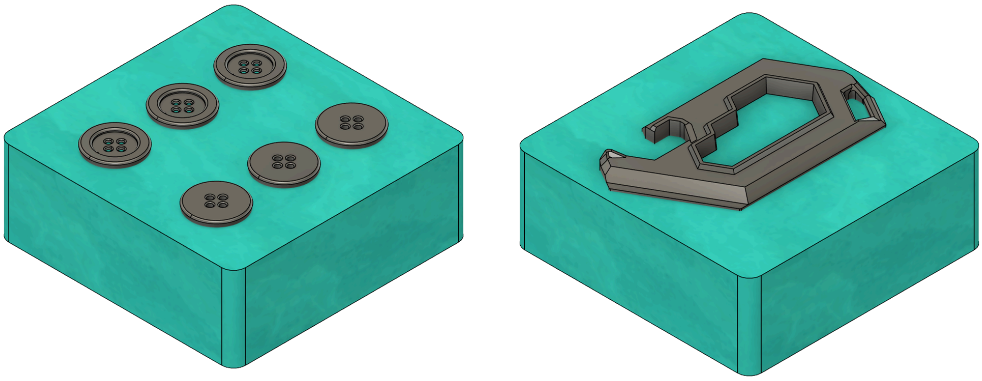
Step 4: Mirror the Inlay Base

Duplicate the inlay base file and mirror it to create the second mould half. Make sure both halves are perfectly aligned on top of each other.



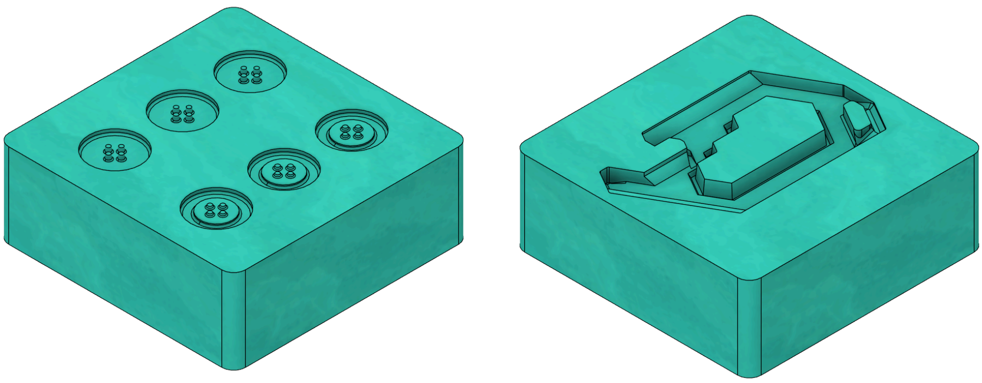
Step 5: Place the Product

Insert the product model into the prepared base file in the desired quantity and position it correctly.



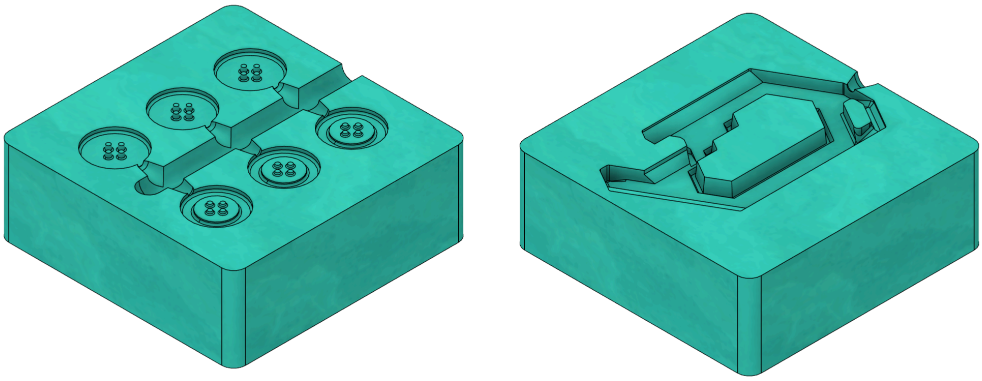
Step 6: Create the cavity

Cut the product model out of both mould halves to generate the negative cavity of the part.



Step 7: Add the Injection Channel

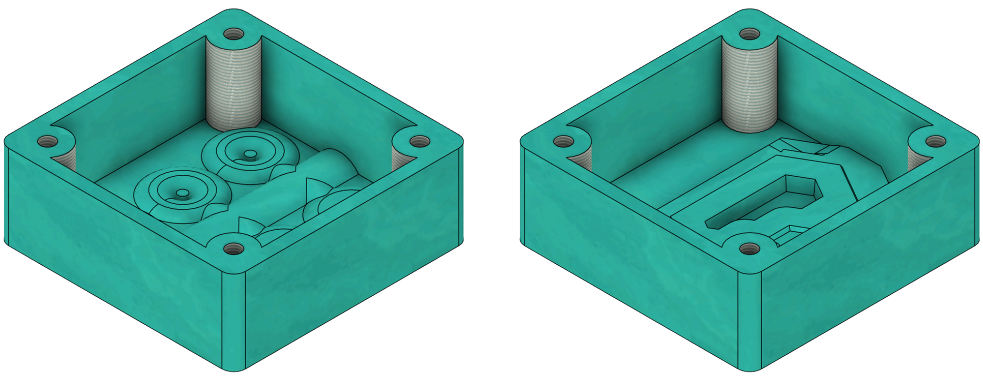
Design the injection channel according to the selected injection method.
The aluminium mould features a **6 mm** diameter inlet — we recommend tapering the channel towards the product to minimize visible gate marks on the final part.



If aluminium inlays are being produced, skip the next two steps and continue directly with Step 10.

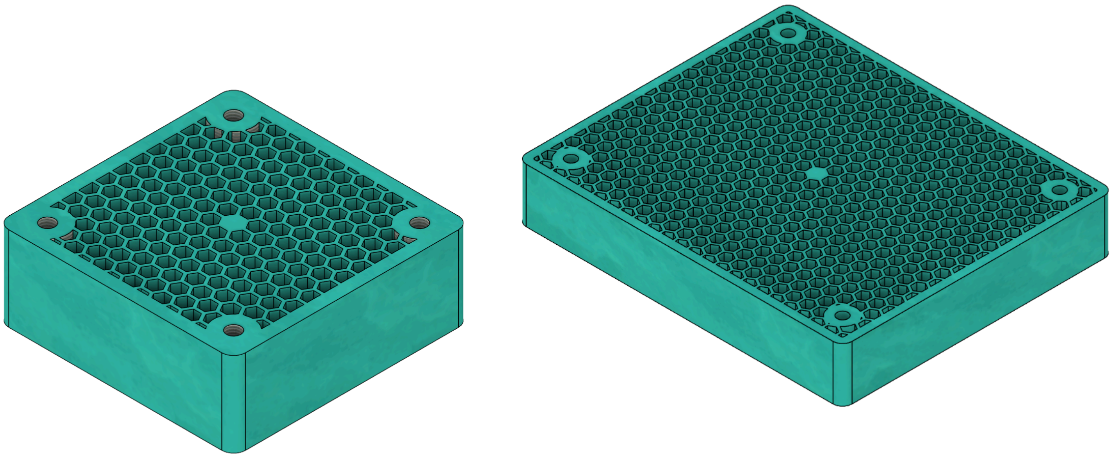
Step 8: Create Material Cutouts – Form a Shell

To save material (especially SLA resin) and to minimize warping of larger inlays, create a shell structure on the underside of the inlay with a **minimum wall thickness of 3 mm**.



Step 9: Add a Hexagon Honeycomb Structure

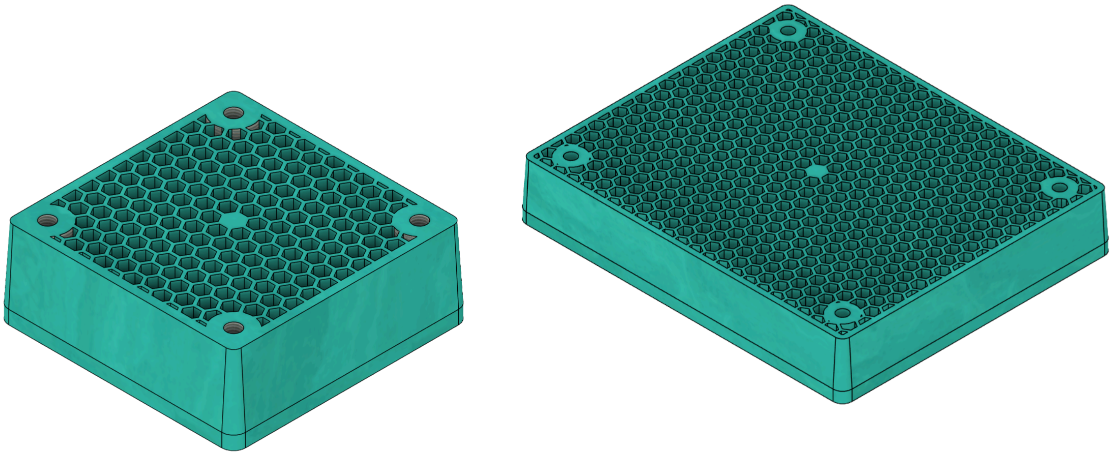
Fill the created shell with a **hexagonal honeycomb pattern** to reduce weight and material usage without compromising the structural integrity.
Make sure that the cavity area is not affected.



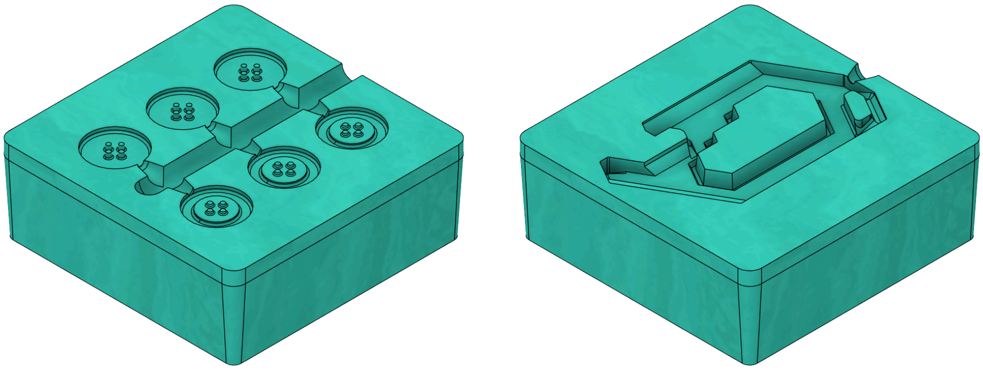
Step 10: Add a Chamfer

Add a continuous chamfer at the bottom edge of the inlay to facilitate easy insertion into the aluminium frame.
The chamfer should have an angle of **3.5°** and a a height of:

- **21 mm** for the 70x70x50 mm variant
- **16 mm** for the 120x100x40 mm variant

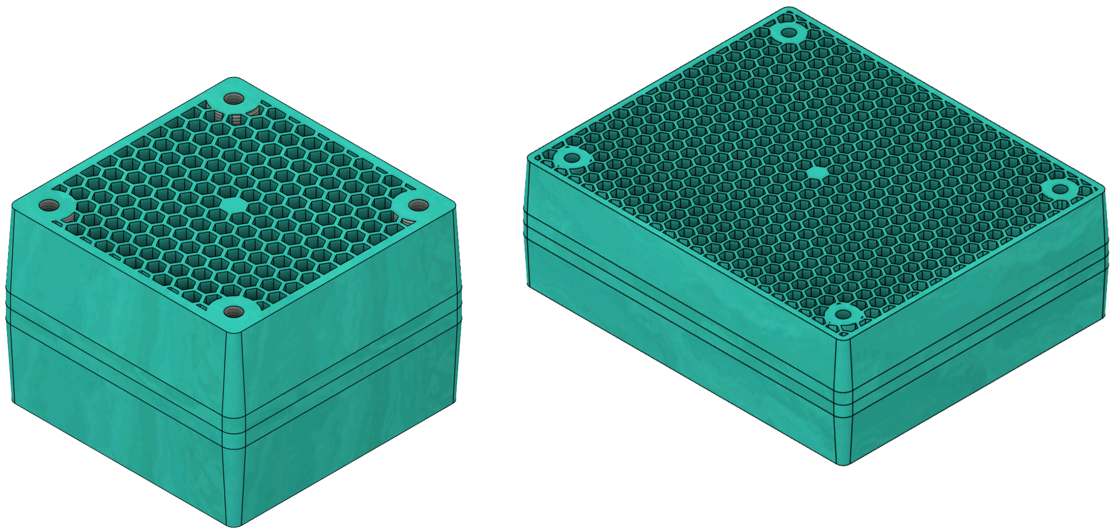


Your Mould Inlay Half is Now Ready for Production



Step 11: Create the Second Mould Half

Repeat the previous steps to design the second half of the mould.
For **symmetrical products**, you can simply mirror the first half — this saves time and reduces the risk of design errors.



Step 12: Manufacturing the Mould Inlays

When manufacturing 3D-printed inlays, keep the following guidelines in mind:

Always use the **smallest available layer height** to achieve the smoothest surface finish and highest level of detail.

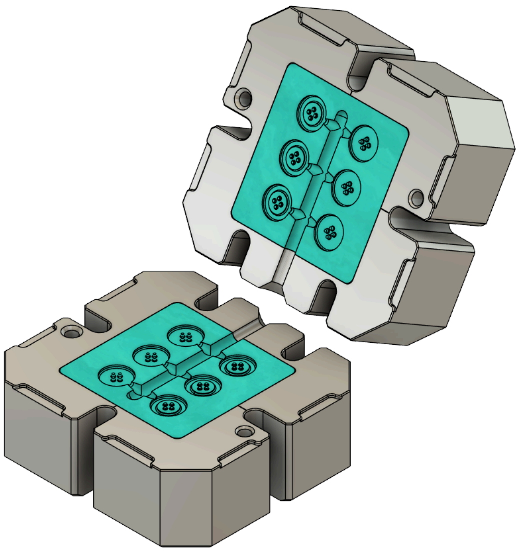
For FDM printing, apply at least the following settings:

- **3 mm perimeter (outer wall) thickness**
- **3 mm top and bottom layers**
- **Minimum 30% infill,** preferably with a strong structure (e.g., gyroid or grid pattern)

Step 13: Installing the Inlays into the Universal Mould

Depending on the accuracy of the 3D printer, the inlays should fit **seamlessly into the aluminium frame.** If the fit is too tight, carefully sand the sides with **fine-grit sandpaper** until a precise fit is achieved.

The inlays are secured with **four M5 countersunk screws**, inserted into the pre-threaded holes provided in the aluminium mould.



Working Guidelines for Using 3D-Printed Mould Inlays

⚠ Always use the **lowest possible processing temperature** to protect the inlay material. Excessive temperatures can significantly reduce the inlay's lifespan.

⚠ **Keep the injection pressure as low as possible** to avoid deformation or damage to the inlay.