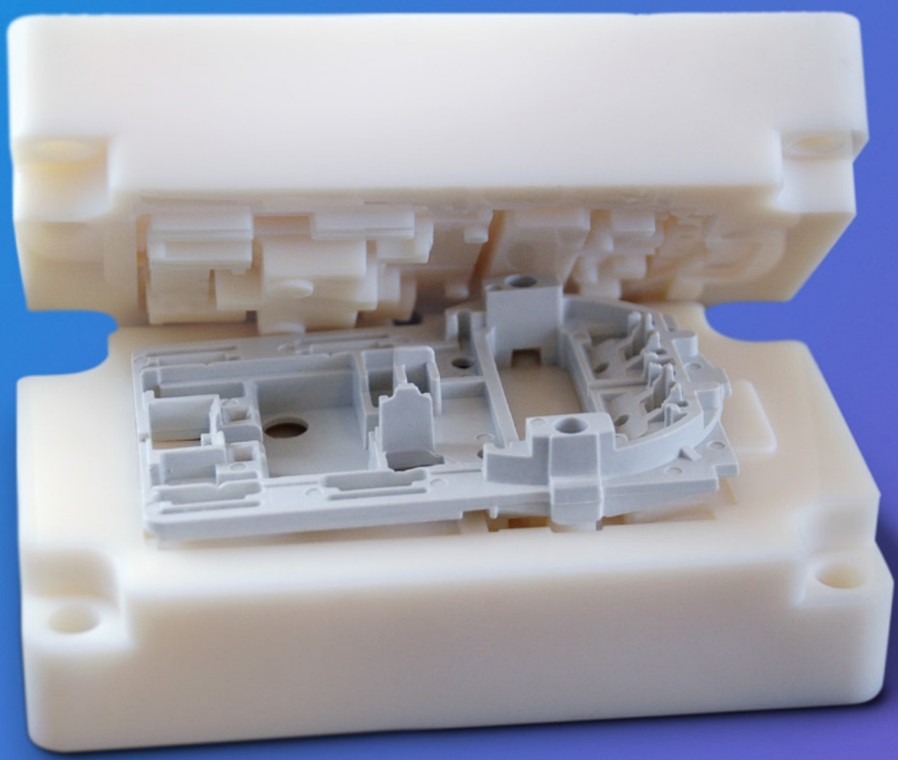


Injection Molding using Rapid Tooling

Somos® Materials

covestro

At a Glance

Tooling is a highly technical application with a number of variables. While we recommend reading through this guide in full, these are some key points to consider. Our team is also happy to assist you.

When should I use rapid tooling?

- Part size is relatively small, less than 127 mm (5 inches).
- Part design still requires iterations.
- Verification of final production design requires molded plastic.
- Quantity is between 1 and 500 parts.

What are mold design guidelines for additive manufacturing?

- Draft angle minimum of 2°.
- Avoid tall thin features that can break during molding.
- Ejector holes (slightly undersized) should be printed into the mold and later machined to the exact size.
- Larger gating should be used to allow for material flow during molding.
- Conformal cooling within the mold is not needed and adds little benefit.
- Design cooling into the metal base plate(s) to control mold temperature and prevent failure due to thermal stress.

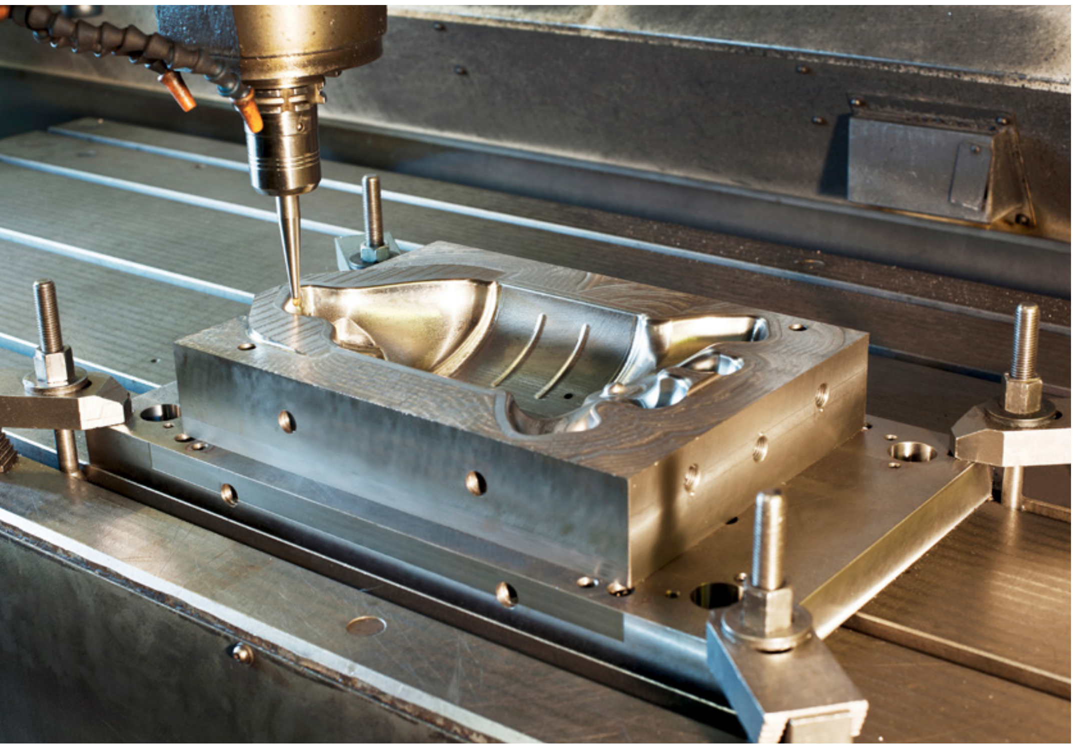
What else should I be aware of?

- Every mold and part is different; care taken to optimize each setup in the beginning of the process will yield the best results.
- Molds will need curved surfaces to be sanded or finished to desired level to remove layer lines and prevent part sticking during molding process.
- Mold temperature controllers should be run at the same settings as for metal tooling.
- Use mold release with every shot.
- Keep the mold open after the part is ejected and blow compressed air onto the core and cavity for at least 30 seconds.
- The cycle time for tools made with Somos® composite materials will normally be between 60 and 120 seconds. Cooling times are dependent on part size, geometry, wall thicknesses and the type of plastic being molded.
- Use the lowest clamping force possible to achieve optimal part results. Start the first shot with lowest clamp force and work up.
- Use a moderate injection speed; start the first shot at lower injection and work up.

Introduction

Testing a new design before costly tooling is created can save companies time and money. For many years, the only process available to create injection molded parts for testing and design verification was "soft-tooling" using machined aluminum. The cost involved in producing machined aluminum tools and the typical lead-times involved are often expensive and can take many weeks.

Until recently, using 3D printing for design verification and product certification was difficult as the resulting parts were made of a different material from the final injection molded plastic. This difference in material would often yield different test results and made it difficult to receive certifications such as UL or CE. With the introduction of composite materials, engineers can now use quick and accurate 3D printing technology to rapidly create molds capable of producing real injection molded parts that can reduce cost and time in the product development cycle.



Contents

Overview

For short run and prototype tooling consider rapid tooling with **Somos®** materials.

Application Details

Injection molded parts in less than three to five days – that's the goal of design houses using rapid tooling with **Somos®** materials and DLP/LCD 3D printing. From there, plastic parts are produced that engineers can test during the product design stage. This is not production tooling, but it does give the design engineer the ability to quickly and cost-effectively obtain hundreds (and in some cases thousands) of parts.

When to Use Rapid Tooling with Somos® Materials

- Part size is relatively small, less than 127 mm (5 inches)
- Part design still needs iterations
- Verification of final design requires production plastic
- Quantity of parts is low (between 50 and 100 parts)

Benefits of Using Rapid Tooling

- Receive actual injection molded parts in less than five days
- Shorten qualification cycle
- Save tooling costs on low production volume runs
- Hand-loaded cores can be used in this method
- Radius capabilities of 3D printed tools are generally better than those made from CNC machining. If a low volume of parts is needed, stereolithography and DLP/ LCD 3D printing provide design freedom that surpasses that of machined molds.

Compatible Plastics

Many commonly used thermoplastics can be used in rapid tooling and more are being qualified.

- Polyethylene
- Polypropylene
- Thermoplastic elastomers
- High impact polystyrene
- ABS
- Polycarbonate
- Glass-filled PA

Mold Design

Taking full advantage of the 3D printing process is critical to unlocking the most benefits from rapid tooling.

Considerations

- Consider using rapid tooling with parts less than 127mm (5 inches)
- Ribs: no less than 1.6mm
- Bosses need to be tapered by at least 3°
- Maximum height to diameter ratio 3:1
- Draft angle minimum of 2°
- Composite materials like **Somos® PerFORM** are somewhat brittle. Tall features and thin-walled details that work well in steel and aluminum molds may fail using a composite resin mold. Features such as these should be made of metal and inserted into the mold.

Photo: Courtesy of Wehl & Partner

Success Example

Wehl & Partner used **Somos® PerFORM** to successfully make nearly 80 ABS parts in just a few days using rapid tooling, without sacrificing detail or accuracy of the plastic part.

Ejector Pin Holes

Stereolithography and DLP/LCD 3D printing provide a high degree of accuracy. For this reason, it is indispensable that ejector pinholes be built in to the molds. These pinholes should be slightly undersized and then later reamed to the exact size.

Gates

- Areas such as gates should be made of steel if wear is anticipated due to the mold design or the projected number of parts. This is especially important if glass-filled plastics will be molded.
- Gates may need to be larger than normal and the mold inserts may need to be more vented.

Draft

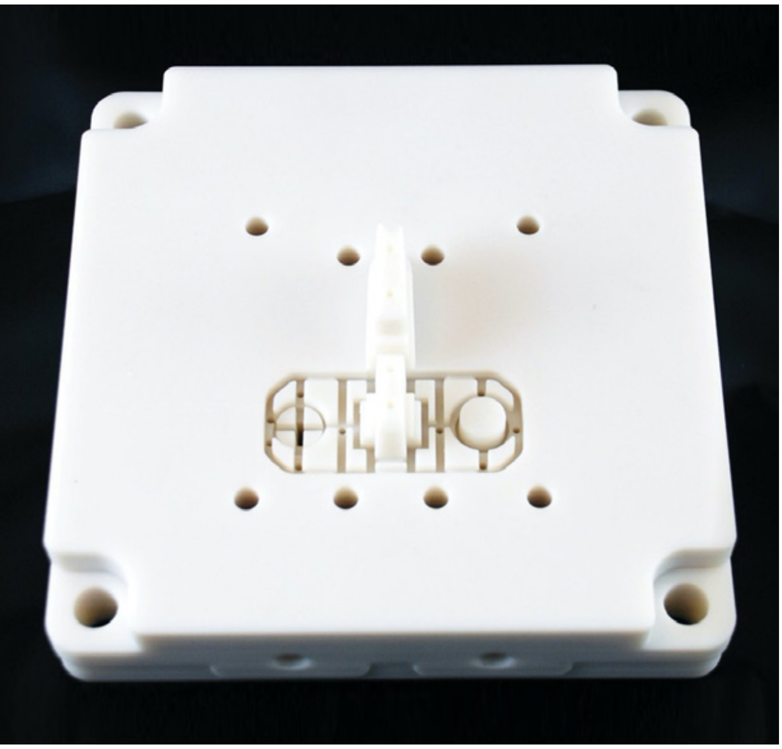
Draft angles may need to be increased. A minimum draft angle of 2° is preferred and, in some cases, more may be necessary to accommodate the proper ejection of the plastic piece.

Reduced Pressure Plane

It may be helpful to design into the mold a reduced pressure plane around the part cavity to help concentrate clamp force which minimizes flash and enhances venting. The plane should be raised 0.2 mm around the part cavity.

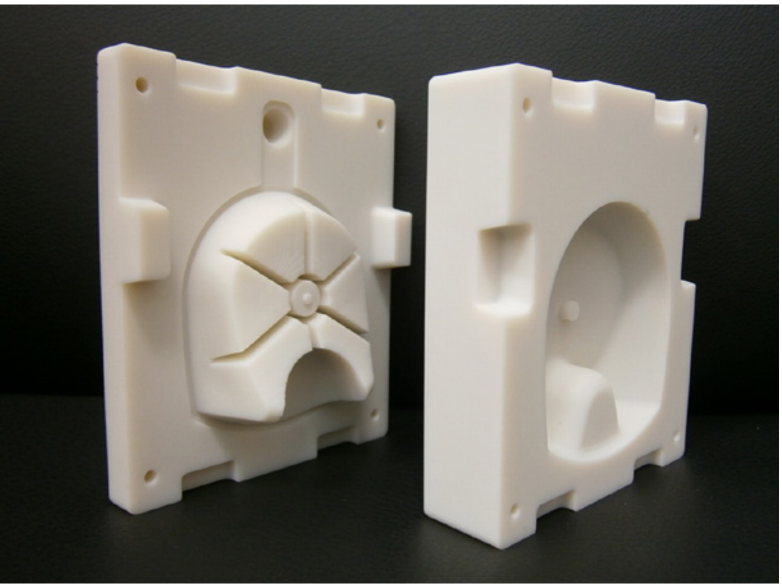
Printing the Mold

Before printing, there are some key items to consider.



Orientation

- Choosing the proper orientation for building a mold is important and will be highly dependent on the mold size and configuration.
- Building mold inserts with the core and cavity facing up (in the z direction) will minimize build time, but may add more finishing time to the process. If the inserts are built in this orientation, it is recommended that one or two layers (0.1 mm to 0.2 mm) be added to the mating surfaces. These layers can then be machined off to ensure complete shutoff and to minimize flash.
- For molds with curved surfaces or non-flat parting lines, it is recommended that the inserts be built standing on edge.
  This will increase the build time, but minimize finishing time.



Layers

Build layers thickness should be chosen based on the required level of detail. The best results have been obtained using a high-resolution mode if equipped. However, most injection molded part requirements can be met using a 0.1 mm layer thickness.

Fit

For use in injection molding machines, mold inserts should be designed to fit into standard steel or aluminum MUD frames. The inserts should be built slightly oversized so that the mounting surface can be machined to an exact fit into the frames.

Photos: Courtesy of Sinthesi Engineering

Preparing the Mold for Use

All molds will require some level of final finishing and surface treatment before they are ready for molding operations.

Machining

- In order to use composite mold inserts, they should be machined to fit tightly into steel or aluminum MUD frames. A tight fit is extremely important to prevent the inserts from expanding and possibly cracking during use.
- Since composite materials such as **Somos® PerFORM** contain ceramic particles, grinding is an excellent method for squaring off the inserts. Machining can also be performed using carbide tools. Standard tools made of steel will also work, but they will have increased wear.
- As **Somos®** materials can be heat treated to increase its heat deflection temperature characteristics and hardness, it may be advised to wait until after machining to perform thermal treatment. This will make machining the part easier.
- As with all materials, proper PPE (personal protection equipment) should be worn when machining, sanding or grinding composites.

Milling

- For milling, it is recommended to use three or four flute end mills. These should be used at high spindle speeds and slow feed rates. This will minimize breakout. Cutting depths of 0.5–0.7 mm are possible under these conditions.
- Oil based lubricants and coolants should be avoided. A mixture of dishwashing detergent and water works well and permits easy cleanup.

Mating Surfaces

The mating surfaces of the mold inserts should only protrude above the MUD frame by 0.2–0.5 mm. Maximum is 0.5 mm.

Finishing

- After machining the mold inserts, they should be sanded and polished as necessary to achieve the required surface finish. It is best to remove all layer lines that could prevent plastic parts from being ejected properly. Commercially available sandpapers work well (use medium-grit wet / dry sandpaper [180- to 220-grit]). Both wet and dry sanding will work, although wet sanding is preferred to keep creation of dust to a minimum. Complete this sanding step with fine-grit sandpaper (320- to 400-grit).
- For polishing, it is recommended to use water-based compounds, as they are easy to remove and leave no residue. A good product for polishing is Wright's Silver Cream. It is readily available in most supermarkets and made by the J. A. Wright Company (www.jawright.com).

Surface Treatment

- A surface treatment with release agents is recommended. Appropriate release agents for the plastics being molded should be used.
- Before molding, release agents should be sprayed a few times into the core and cavity, and if needed after every shot.
- For some plastics, it is recommended that the molds be surface treated with silicone oil and thermally cured. Although not always needed, this has been found to greatly reduce the chance of parts sticking in the mold. Surface treatment using silicone oil is not recommended for PE, PP or thermoplastic elastomers. For further information on surface treatment, refer to the section in the appendix of this guide titles "Extending Tool Life."

Molding Process

Slight adjustments to the molding process can be used to increase the working life of the mold and produce the most parts.

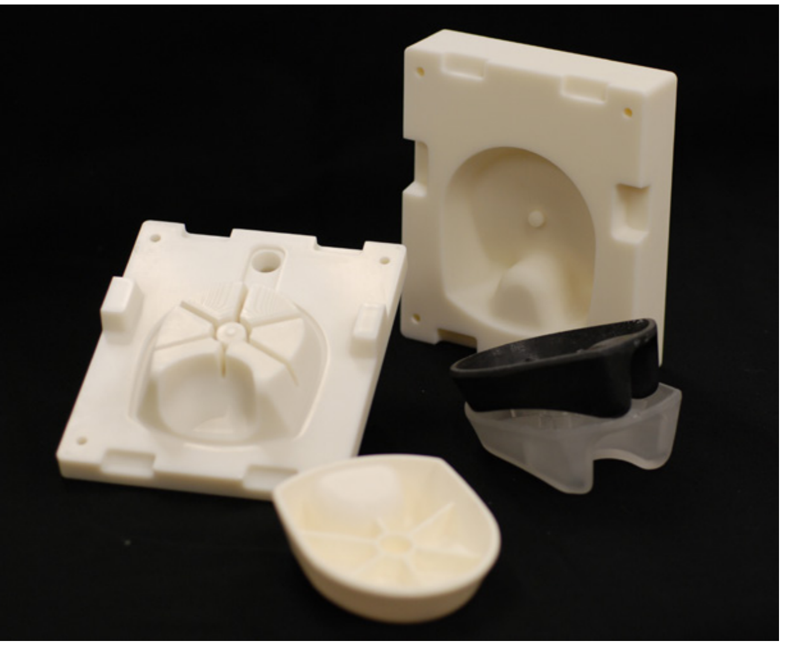


Photo: Courtesy of Sinthesi Engineering

Cycle Times

- Since the thermal conductivity of composite materials is considerably lower than that of steel or aluminum, it does not dissipate heat as quickly as a metal.
- The cycle time for tools made with **Somos®** composite materials will normally be between 60 and 120 seconds. Cooling times are dependent on part size, geometry, wall thicknesses and the type of plastic being molded.
- Typically, cycle times are longer than seen in conventional steel or aluminum tools, but as this process is intended for short runs and not for large production volumes cycle time is not as important as the overall turnaround time. This method is intended to quickly and cost effectively produce actual injection molded parts within days in quantities ranging from 1–100, and in many cases even more.

Controlling Heat Buildup

- As **Somos®** materials are highly heat resistant, mold temperature controllers should be run at the normal setting used as for metal tooling.
- Keep the mold open after the part is ejected and blow compressed air onto the core and cavity set.
- Heat buildup can also be controlled by reducing the temperature of the mold and increasing the injection pressure. This should be done carefully to prevent damage to the mold. Start with 50% of the pressure used with an aluminum tool and increase incrementally.
- Cooling channels will not dramatically help regulate temperature within the mold since the low thermal conductivity of the composite does not dissipate heat. This is especially true of thicker molds. Cooling channels may help with thinner molds.

Pressures and Clamping

- Use the lowest clamping force possible to achieve good part results and reduce the chance of damaging the mold.
- Limit the maximum molding machine pressure to avoid pressure peaks during fill.
- Use a moderate injection speed.

Appendix

Extending Tool Life

Materials such as **Somos® PerFORM** are highly filled materials that offer several advantages over standard resins, including low shrinkage and high heat resistance. As such, these composite materials are suitable for injection molding tool applications. In order to achieve the best mold release properties and extend the working life of the tool, it is recommended in most cases that a surface treatment should be applied after the final finishing of the tooling or insert has been completed. The surface treatment can be the use of mold release agents, sprayed numerous times into the core and cavity, and in between shots, or a more aggressive treatment with silicone oil can be used. If silicone oil is to be used, it will need to applied after the mold has been cleaned and UV post cured by the service provider. The following outlines the procedure for applying and curing the silicone oil.

Surface Treatment Procedure

- Wash the parts for no more than 30 minutes, including drain time, with a glycol ether such as DpnB, TPM or DpnP. Efficiency is often increased by using an ultrasonic cleaner during this process.
- Rinse the parts with isopropyl alcohol to remove the glycol ether. It is best to not leave the part in these solvents for too long due to their degradation in these conditions if they have not been UV post-cured.
- UV post-cure the parts for one hour (30 minutes per side).
- With a brush, apply liberal amounts of silicone oil to the tooling surface. Be sure to cover all areas, especially inside the corners, ribs, bosses, etc.
- Place parts in a 40°C oven for one hour.
- Wipe all excess oil off the part surface. Allow to cool to room temperature and clean with isopropyl alcohol, ethanol or MEK. All molds or inserts must then dry at room temperature to evaporate solvents.
- Place the parts in a programmable oven, using the following cycle:
  1) Warm up: Place the molds in the oven and ramp the temperature from room temperature to 150°C over two hours.
  2) Hold at 150°C for two hours.
  3) Cool down: Ramp the oven from 150°C to room temperature over two hours.
- Special note: Parts with thin walls and/or unsupported overhangs can distort during the thermal treatment process.
  1) Parts with these geometries should be supported properly to post-cure. If standard fixtures are not available, a container of glass beads may be used that are approximately 3-4 mm diameter. Immerse the parts to be thermally post-cured into a rigid container of glass beads. The container should be able to withstand the thermal cure process. The top of the beads should be weighted down; this will minimize the molds movement during the thermal cycle.
  2) The use of glass beads adds mass, requiring more time to heat up and cool down. If glass beads are used, insert a thermocouple into the center of the beads. Proceed with the normal two hour warm up, but wait until the thermocouple reads 150°C before starting the hold stage. Hold at 150°C for two hours and then proceed with the normal two hour cool down process. Wait for the thermocouple to read 35°C before removing molds.

Questions

For questions about rapid tooling or any of the processes outlined above, contact Covestro. Please also consult the printer manufacturer print guidelines and settings.

More information at am.covestro.com

Covestro Deutschland AG
Kaiser-Wilhelm-Allee 60
51373 Leverkusen
Germany

www.covestro.com

Edition: September 2021 · Printed in Germany