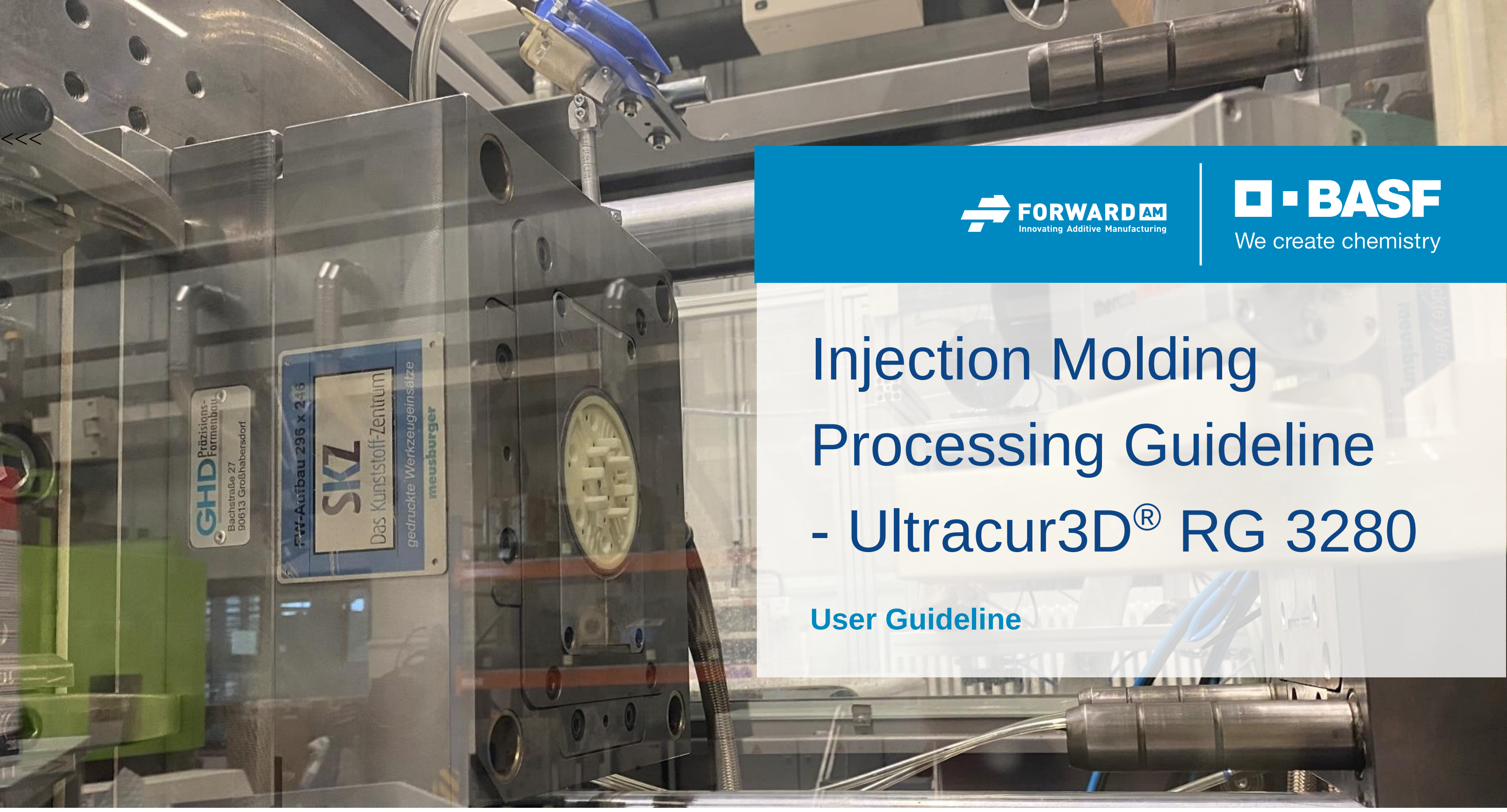


FORWARD AM Innovating Additive Manufacturing

BASF We create chemistry

Injection Molding Processing Guideline - Ultracur3D® RG 3280

User Guideline

DESCRIPTION

3D printed mold inserts are mainly used in prototype toolmaking (rapid tooling) but can produce near-series functional parts of low to medium quantities (approx. 100-1000) as well. This is especially useful to quickly identify critical part geometries during mold and end use part development.

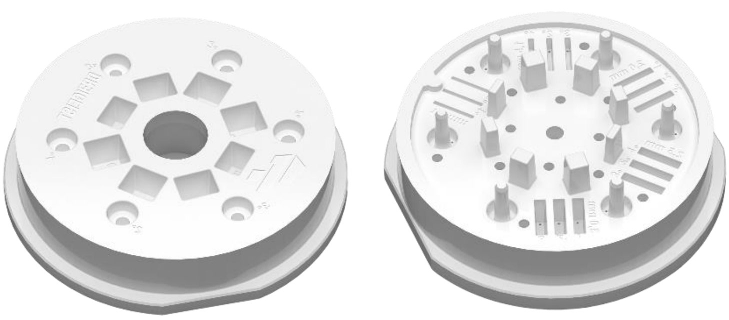
QUICK FACTS

Material:
Ultracur3D® RG 3280

Technology:
Digital Light Processing (DLP)

Partner: DREIGEIST Additive Intelligence

The use of printed mold inserts for the injection molding process requires careful preparation of the molds to be printed as well as focused attention to the individual steps required to move seamlessly from design to printing, post-processing and pre-treatment to the actual injection molding process.



"Stonehenge V3 reference design" STL file by SKZ & DREIGEIST

Source: DREIGEIST

To achieve the best possible performance, the following chapters should be followed.

Based on the test series carried out at SKZ – German Plastics Center, together with our partner DREIGEIST, it was possible to determine basic guidelines based on the design on the left.

Version 1.0

White, ceramic-like color & feel

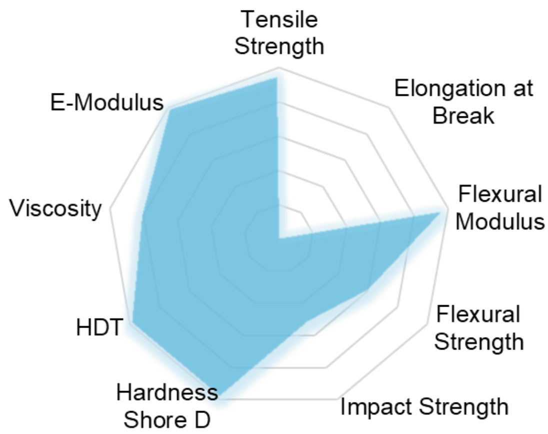
Perfectly suitable for tooling & molding

Extended technical data

MATERIAL DETAILS

Ceramic-filled resin with exceptionally high stiffness and temperature resistance

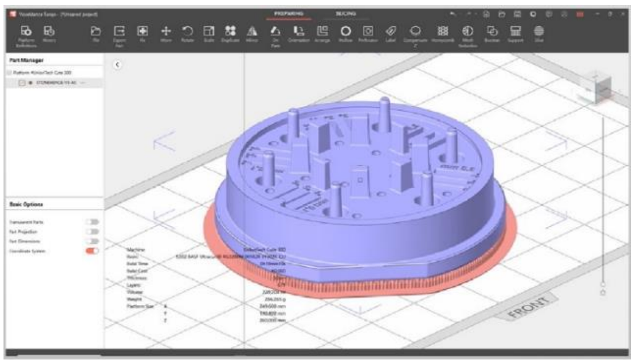
- Superior stiffness: 10 GPa
- Superior temperature performance: HDT > 280°C
- Very fast and easy to print
- High suspension stability
- Exceptional chemical resistance
- TPO-free



PRINTING

Ultracur3D® RG 3280 is compatible with multiple 3D printers. However, a high-quality light engine is preferable to achieve the recommended layer times and resolution. An overview of validated 3D printers can be found here: **Ultracur3D® RG 3280 – Printer Compatibility**.

It is recommended to orient the molds in a way that achieves the best possible edge sharpness at the parting line; meaning the molds are printed either horizontally or vertically to the printing plate. Overhangs and undercuts should be avoided, otherwise a support structure will be required within the cavity leading to quality losses. If ejector holes and/or mold vents are present, a horizontal printing orientation is preferred. In this case, a support structure (see picture, orange section) is required for a better material flow and to avoid vacuum creation at the holes, which can lead to quality reduction or print failure.



Source: DREIGEIST

To achieve the full potential of detail sharpness the preferred layer thickness would be between 50 and 70 microns, though layer thicknesses of up to 100 microns are also feasible. For massive geometries, as is the case with mold inserts for injection molding, longer travel distances and longer shift waiting times are crucial. A fully flat surface should be ensured, especially at the parting line and also on the back of the molds, if necessary utilizing mechanical finishing. Otherwise, this can lead to stress fracture when clamping in the molds.

POST PROCESSING

To achieve the final mechanical properties and necessary thermal resistance of the molds, post processing is required after printing. The individual steps are illustrated below:

In the **User Guideline**, initial material settings and handling recommendation can be found. Heat treatment is strongly recommended as it can increase heat resistance, resulting in higher durability and preventing the expected shrinkage under thermal stress during the process.

ADDITIONAL TREATMENT

The tests pointed out that the durability of the molds could be significantly improved by additional treatment. Different release agents were tested. Following treatment was used with wet release agents which improved the overall demoldability, as well as prevented frequent reapplication thereof: The molds were sprayed on either side of the cavities and treated in a heating oven at 60° - 70°C for 16 - 24 hours. Dry release agents were applied after every 5-15 injections.

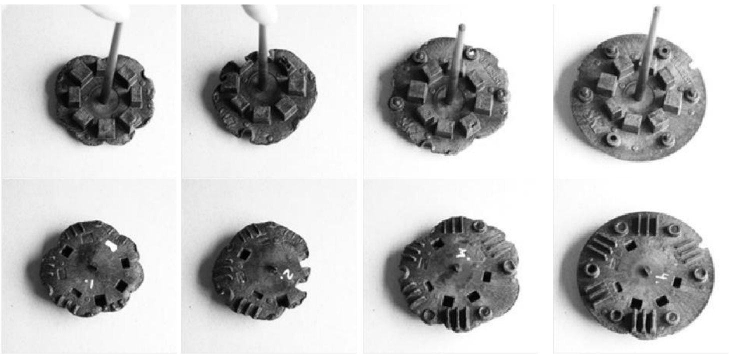
The following release agents showed demoldability improvements in the order as listed below:

- Silicon Oil (Ballistol)
- Silicon-free release agent “020” (Jost Chemical)
- Dry release agent “PC EXTRA” (Jost Chemical)
- PTFE-Spray “FT-300” (Jost Chemical)

All release agents have shown a positive effect, with the silicone oil and the silicone-free release agent (020) being recommended. Since the use of release agents is not limited to the once tested, there is nothing to prevent trials with release agents used in-house. As mentioned under **PRINTING**, a flat surface is necessary so that no stress fracture occurs when clamping the inserts (screwing, clamping, etc.). If this is not feasible, the printing strategy (printing parameters and/or orientation) must be checked and optimized. Alternatively, the molds could be reworked mechanically, but the material brittleness needs to be considered. Milling and grinding usually work well, keeping in mind that the material is brittle and therefore a very low material removal rate (about 0.1 mm) at a moderate to slow travel speed is recommended.

forward-am.com +49 6221 67417 900 sales@basf-3dps.com

INJECTION MOLDING PROCESS



Starting with 70% filling of the mold

Source: DREIGEIST

For injection molding it is recommended to start with a filling study, as this also helps to bring the mold into thermal balance before full loading forces are applied. It is advisable to start with lower forces such as closing and locking forces, injection pressure, etc., but to also avoid excessive over molding.

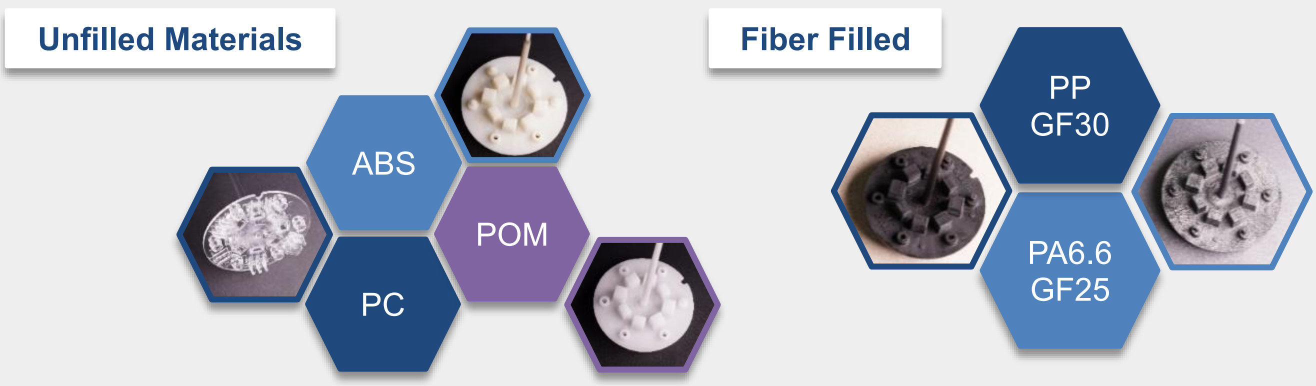
Among other things, a soft ramp should be set for closing and opening the tool halves. The ejectors should also demold the molded part with less force and speed.

An integrated cooling of the molds is conceivable but has not yet been tested. Plastic is a thermal insulator hence active cooling, such as the compressed air used in the described trials, is required.

Due to the high Tg temperature of the Ultracur3D® RG 3280 (see **extended TDS**), higher mold temperatures and demolding temperatures are possible, so that optimization of cycle times is also feasible. However, this can lead to a higher wear rate, which would reduce output. In order to achieve the best possible service life and molded part quality, a longer cooling time is recommended.

The setting parameters for the injection molding process should be based on the material supplier's specifications (process instructions).

Different injection materials have been successfully tested:



The following setting parameters have been established for the process and can be considered as guiding values. The data is based on the tests carried out and are intended only as a guide. To design your process optimally, the characteristic values from the guidelines and process instructions of the material manufacturer (for the injection materials) should be used while also considering varying requirements of the particular injection molding setup.

Based on	Screw diameter 30mm	PP GF30	ABS	PA6.6 GF25	POM	PC
Speeds	Circumferential speed (mm/s)	250	250	250	250	250
	Screw feed rate(cm^3/s)	30	30	30	30	30
Paths	Switchover point to p_N (cm^3)	7,0	5,5	11,3	7,3	7,7
	Dosing path (cm^3)	28	28	30	30	30
	Mass cushion (cm^3)	3,1	2,1	5,6	6,2	6,1
Times	Cycle time (s)	138	160	90	110	169
	Injection time (s)	1,06	1,12	1,2	1,1	1,1
	Post-pressure time (s)	6	7	6	6	5
	Residual cooling time (s) (In-mold + air pressure cooling)	60 + 60	80 + 60	40 + 35	60 + 35	90 + 65
	Dosing time (s)	4,4	4,1	5,9	3,9	4,2
Pressures	Injection pressure (bar)	530	600	670	920	830
	Holding pressure (bar)	190	200	130	200	200
	Back pressure (bar)	60	60	60	60	60
Temperatures	Feeding zone (°C)	30	30	60	30	70
	Cylinder temperature 1 (°C)	200	210	245	170	270
	Cylinder temperature 2 (°C)	210	220	255	180	280
	Cylinder temperature 3 (°C)	220	230	265	190	290
	Nozzle temperature (°C)	220	230	290	190	290
	Temperature control unit Nozzle side (°C)	30	30	90	60	60
	Temperature control unit Ejector side (°C)	30	30	90	60	60
Forces	Clamping force (kN)	300	300	300	300	300
Drying	Drying time (h)	-	4-6	4-6	4-6	4-6
	Drying temperature (°C)	-	80	80	80	120

Version 1.0

MOLD DESIGN

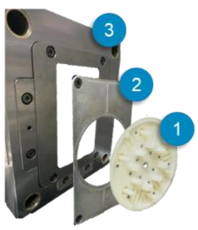
In general, the design guidelines of injection molds should be applied, but with the consideration that thin features should be avoided, since the mold made of a ceramic printed material such as the Ultracur3D® RG 3280, has lower strengths than aluminum or steel molds.

It is also advisable to design the molds as inserts fixed in a master mold to avoid large components and print only the most necessary area. Experience has shown that a size of up to 100 x 100 mm with a maximum thickness of 50 mm is most suitable. Larger molds are certainly feasible, but care must be taken here to ensure that they are warp-free.

Important notes for Mold Design:

- Uniform wall thickness to ensure proper flow of molten plastic
- Draft angles of at least 2° - 3° for amorphous and 3° - 5° for semi-crystalline material on vertical walls to allow for easy demolding
- To reduce wear, the area opposite the injection point is recommended to be reinforced with meal inserts or an ejector should be placed

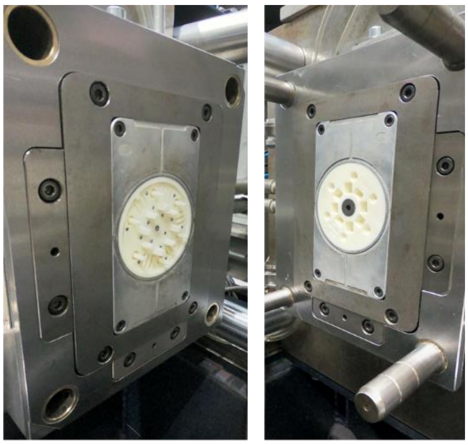
INJECTION MOLDING PROCESS SIMULATION



1 3D printed insert

2 Aluminum mold

3 Master mold



3D-printed insert in base mold

For simulation of the molding process with e.g. Moldflow® the difference in thermal property of a 3D-printed mold compared to a metal mold must be considered (see table below). We recommend modeling a 3D-printed insert within a master metal mold utilizing the material data from our **extended TDS** or a material card for Moldflow® can be provided on request.

Mold Material	Conductivity @200°C W/(m*C)	Spec. Heat @200°C (J/(kg*C))
Ultracur3D® RG 3820	0.69	1810
Aluminum	190	880
Mold-Steel	29	460

We can support your project with years of simulation experience in injection molding. If you have any questions or would like more information, contact: sales@basf-3dps.com.