# P3™ Silicone 25A Gray
## Material Processing Guide

ORIGIN

This guide provides information for processing P3™ Silicone 25A Gray by Shin-Etsu.

### Profile

**In GrabCAD Print™, select the *Elastomeric* or *Elastomeric - Conservative* default support profile.**
The conservative support profiles create more contact points between the model and the support structure, thereby increasing print success rates.

### Cleaning

| Cleaning Steps* | Duration |
|---|---|
| 1st bath of **IPA** (ultrasonic) or **Acetone** (swirl-based agitation)<br>(Rinses off excess resin from the surface, but accumulates dirt quickly) | 2 minutes |
| Dry parts with compressed air. | 10–60 seconds |
| 2nd bath of **IPA** (ultrasonic) or **Acetone** (swirl-based agitation)<br>(Final rinse of the part, remains clean longer than the first bath) | 2 minutes |
| Dry parts with compressed air. | 10–60 seconds |
| Rinse part using Acetone squeeze bottle and let it dry. | 10–60 seconds |

* If a part is not fully clean (still appears wet after compressed air treatment with no evaporation), repeat the 2nd bath, followed by compressed air drying and an Acetone rinse.

### Wait

**Wait at least 60 minutes.**
If the parts require a total sonication time exceeding 5 minutes, let the part dry for at least 3 hours.

### UV Curing

Use the UV curing system for your printer.

**Origin Cure™/CureLite™** (Origin Two/Origin One+)

- Place parts.
- Select the material program.
- Tap **Start**.

**Dymax ECE5000** (Origin One)

- Place parts.
- 15-20 minutes per side
- Shelf: I

### Thermal Curing & Humidity Control

| Phase | Duration | Temperature |
|---|---|---|
| Ramp up | ~30 minutes | 22°C to 85°C (3°C/min)<br>85% RH |
| Hold | 18 - 24 hours | 85°C / 85% RH |
| Oven off | Until cool down | To room temperature (RT) |

**Recommended**: Create a thermal curing program for this step.

If the oven does not have humidity control, place a large water tank inside to increase humidity in a well-sealed oven. This setup helps maintain high humidity levels, preventing excessive drying of printed silicone parts and ensuring a proper post-cure process.

# Material Overview

P3 Silicone 25A Gray is a soft, flexible, and durable elastomer designed for applications requiring excellent tear resistance, elongation, chemical resistance, and consistent mechanical performance. With a Shore A hardness of 25, this material offers a balance of softness and strength, making it ideal for producing seals, gaskets, selected medical and wearable applications, and parts for consumer goods. Its high resilience and stability across a wide temperature range ensures reliable performance in industrial, automotive, aerospace, healthcare, and consumer products. Silicone 25A Gray requires proper handling and post-processing to achieve optimal properties and surface finish.

- For details regarding material types, model selection for additive production, and optimizing accuracy and throughput on Origin printers, refer to the P3 Origin Operations and Applications Guide.
- To learn more about the accessories related to the P3 Origin printers, refer to P3 Origin Accessories Product Essentials.

P3 Silicone 25A is a Validated material in GrabCAD Print that passed basic reliability testing.

Table 1: Material features

| Feature | Value |
|---|---|
| Color | Gray |
| Maximum unsupported overhang length | 3 mm |
| Maximum span length | 7 mm |
| Minimum unsupported overhang angle | 15° |
| Minimum vertical wire diameter | |
| 1 mm height | 0.5 mm |
| 3 mm height | 1 mm |
| 5 mm height | 1 mm |
| Minimum unsupported wall thickness | |
| 5 mm height | 0.5 mm |
| 10 mm height | 0.5 mm |
| Minimum hole diameter in Z | 2 mm |
| Minimum hole diameter in XY | 2 mm |

Figure 1: Example Silicone model

# Material Processing Guidelines

## Pre-print Preparations

- To minimize overhangs and reduce excessive support, optimize print orientation.
- The validated settings work well for most geometries. However, some cases may require modifying settings, as follows:
  - For large cross-sectional areas, you may need to increase *Exposure Duration* (in GrabCAD Print) or *Advancement Delay* for OpenAM users.
  - For optimal print quality of large overhangs, consider increasing *Exposure Duration* (in GrabCAD Print).
- To improve adhesion and reduce the need for support, print parts directly on the build platform, when possible.
- To minimize parts sticking to the tray sheet instead of the build platform, in GrabCAD Print, under *Tray Settings* > *Material Properties*, increase *Exposure Duration* and adjust the build platform adhesion settings.

## Support Considerations:

- P3 Silicone 25A is a soft elastomer with low green strength and stiffness.
  - Parts printed with P3 Silicone 25A need more robust support structures in order to print successfully.
  - The low green stiffness of P3 Silicone 25A makes printing tall parts with large aspect ratios challenging.
    To solve wiggle-line defects for tall parts or features with large aspect ratios, strengthen the support structure or change print parameters. This increases the *Advancement Delay*, *Exposure Duration*, and/or make approach settings more conservative.
- P3 Silicone 25A Gray has moderate adhesion to the build platform. Consider adding a brim/raft to parts with small cross sections to improve adhesion.
- The contact point diameter for support should range from 1.2-1.8 mm (1.3 mm is typical).
  - Contact point spacing of 1.5-2.5 mm is typical for flat downward facing surfaces.
  - Tighter spacing is useful for flat surfaces.
  - Angled surfaces can tolerate wider spacing.
  - Note that large tips cause support marks.
- For optimal first print success, consider over-supporting your first print (Conservative profile), then optimizing supports iteratively.
- To ensure support structures remain stable and intact during the print process, design them with adequate thickness and/or structure.
- The optimal support design and type(s) for each print are dependent on part geometry and size.
- To minimize overhangs and ensure better structural integrity, position parts so that the bulky sections or critical features are as low as possible along the Z-axis (closer to the build platform).
  This helps prevent wiggle-line defects.
- The current support profile recommended is Elastomeric support.

# Print Setup

- For optimal print quality, ensure the printing environment is protected from air circulation and temperature fluctuations, as silicone materials can be sensitive to environmental changes, affecting print quality.
- Resin Tray and Build Platform Preparation:
  - To achieve optimal detail and quality of the small detail, adjust printer intensity and resin mixing, as necessary.
  - To prevent cross-contamination and ensure optimal print quality, use a dedicated build platform and resin tray exclusively for P3 Silicone 25A Gray.
    Make sure to use the removal tooling and other jigs exclusively for P3 Silicone 25A Gray.
  - Ensure the resin tray and build platform are thoroughly cleaned and dry before each print.
- Resin Preparation:
  - To ensure a longer shelf life, store the material at ambient temperature or in a cool environment. While refrigeration is possible, before proceeding with printing, ensure the ambient temperature has stabilized.
  - Preheat P3 Silicone 25A Gray to 40°C (104°F) to reduce viscosity, improving flow.
  - Before using the resin bottle, manually mix the bottle for 60 sec or put it on a roller for at least 30 minutes to ensure homogeneity.
  - To prevent inconsistencies, ensure the resin is thoroughly mixed before printing.
    - Overmixing can cause bubbles.
    - Make sure you do not mix forcefully.
- Print temperature: 40°C (104°F)
  This ensures that the resin is not too viscous without compromising the flow.
- Irradiance: 5 mW/cm²
- ProAero™ Air Extractor is not recommended for use with this material. However, if you choose to use an air extractor, set it to the lowest power level.

# Part Removal

- You can remove support structures as follows:
  - **Before or after cleaning**, depending on the support design and part geometry. After cleaning, supports might absorb solvent, making them weaker and easier to detach. However, cleaning—regardless of when it occurs—can also impact the solvent's lifespan.
  - **After all post-processing stages**, if deformation is a concern—keeping the supports in place can help maintain part integrity.
  - To remove support with minimum marks, use a sharp knife or gentle cutter.

Figure 2: Cutter for removing support

- For easier cleaning of parts, allow the completed print to remain on the build platform in the build chamber with the door closed for 15 minutes to allow excess resin to drip off the part.

> **Warning: Hot Surface Hazard**
> The build platform can reach a temperature of 40°C (104°F) and can cause injury. Always allow the build platform to cool or wear insulated heat-resistant gloves to handle the heated build platform when removing it from the printer.

- Place the build platform with the printed parts on a non-reactive portable surface and use a no-drip jig to avoid dripping resin when performing the transfer from the printer onto the non-reactive portable surface.
- Use the Origin scraper tool (ORIG-00305), a plastic putty knife, or gently remove the parts with your hands, which is helpful for detaching the part and supports from the build platform and prevents damaging the part. Slowly work the tool between the print and the build platform with gradual, careful movements.
- For parts with large cross-sections, apply careful, controlled pressure to detach them from the build platform.
- Always push the scraper away from your body, maintain proper blade direction, and keep hands clear of cutting edges.
- Avoid prying with force, as this can deform/scratch soft models. Instead, work around the edges gradually.
- Apply IPA on the edges of the print to help weaken adhesion and then use a spatula to gently pry and remove the part from the build platform.
- Avoid using the corners of the scraper, as this may damage both the part and the build platform coating.

## Cleaning

- Before cleaning, remove support structures from the printed part unless they are needed for structural stability. In that case, wash the part thoroughly.
- Cleaning your print is a 5-step process, which includes two cleaning baths—either in a sonicator or using a swirl-based agitation if using acetone. After each bath, dry the part with an air compressor, followed by a final rinse with acetone.
- Wash parts in two separate solvent baths: first in a “used” IPA or acetone bath followed by a “clean” bath using clean 99% IPA or acetone.
- Keep dedicated solvent containers for each material. Containers should close with an air-tight seal to prevent IPA evaporation.
- Between cleaning baths, use compressed air to dry parts, being especially gentle around delicate features. Always spray machinery or other parts away from the body.
- Inspect the part after the second bath and rinse with acetone.
  If it is not fully clean (still appears wet and evaporation is not evident), repeat the clean bath, subsequent air drying, and an acetone rinse.
- Always aim to minimize the part’s exposure to solvent during cleaning. All materials absorb solvents in their green state to some degree and minimizing time in the solvent reduces the impact on mechanical properties.
- Be careful and avoid damaging delicate features. Aim compressed air away from you, ideally into an air cleaning cabinet or a trash can.

- Negative features, interior corners, and blind holes may be difficult to clean. A cotton swab soaked in IPA can be helpful.

# Post Processing

> **Important:**
> Wait at least 60 minutes after drying before post cure.

- For parts cleaned with sonication for 5 minutes or longer, extend the drying time to a minimum of 3 hours.
- This material requires a two-step post-curing process to ensure the best possible mechanical and thermal properties. This process is divided into UV curing and thermal curing (with humidity control) steps.
  1. UV Curing
  2. Thermal Curing & Humidity Control
- In certain cases, to prevent deformation, support structures should remain intact throughout the entire post-processing procedure.

## UV Curing

- Stratasys recommends Dymax ECE5000, Origin Cure, Origin CureLite, or an equivalent curing solution provided by Stratasys (see the UV Curing table on page 1).
- Flattening with Weight—If the model has a flat geometry, place a glass substrate on top with a weighted object to minimize warpage.
- Use the curing system for your printer:
  - **Dymax ECE5000**—Expose the part in the Dymax ECE5000 for the specified time per side. Thin-walled parts may warp during the post cure process. To avoid this, post cure for shorter intervals, flipping part between exposures. Begin with a 10-second exposure per side to build initial strength before longer intervals.
  - **Origin Cure** and **Origin CureLite**—Use the relevant program.
- Select the correct program according to the material used for printing.
- Place parts on the glass plate and follow these guidelines (For more information, refer to P3 Origin Operations and Applications Guide):
  - Ensure all parts are within the effective curing area.
  - Orient parts properly to avoid deformation and minimize overhanging parts.
  - Spread out the parts uniformly and leave sufficient gaps between parts so they don't touch each other.
  - Place large parts close to the center of the glass plate to minimize shadows on the smaller parts.
  - Tap **Start** and wait for the countdown to finish.
- If parts are still tacky after curing, ensure proper humidity and increase curing duration, if needed.

## Thermal Curing and Humidity Control

- Optimizing Oven Setting—Ensure uniform heat distribution within the curing chamber and use a calibrated temperature profile to reduce thermal stress on the printed parts.

- Using Controlled Cooling—Gradually cool the parts after post curing, instead of exposing them to sudden temperature changes, which can contribute to deformation or thermal shock.
- To limit the warpage for thin flat parts, add a weight on top of the part for thermal post cure.
  Consider keeping the support structure for post-curing or designing a sacrificial stiffening structure for the post-cure process.
- Post-cure cycle for Thermal Curing and Humidity Control:
  Place the parts in an oven (e.g., Memmert™ Humidity Chamber HCP) with the following heating profile:
  - Heating ramp-up 3˚C/min (37.4˚F/min) to 85˚C (185°F) and 85% RH.
  - Isothermal for 18 - 24 hours at 85˚C (185°F)/85% RH.
  - Oven off until cool down to RT.
- If Thermal Curing and Humidity Control are not possible, follow this procedure:
  1. Place dried parts at room temperature oven (around 25°C (77°F)) with 5-liter water tank (1.3 gallon) (minimum 3 liters water will be enough [0.8 gallons]).
  2. Start heating oven with 3˚C/min (37.4°F/min) ramp to 85°C (185°F).
  3. Hold at 85°C (185°F) for 24 hours.
  4. Turn off the oven and keep the door closed until it cools to room temperature.

Figure 3: Thermal cure oven with water tank and Silicone parts

## Post-Printing Maintenance Tasks

- Performing proper maintenance and cleaning up after failures significantly improves print consistency and reduces the risk of recurring defects.
- After a print is completed (and no other prints are planned), turn off the heating and cover the resin tray, as the silicone is sensitive to humidity and heat.
- After a print failure, in the printer tap **Full Area Projection** (at least 30-45 sec) to remove any remaining support structures or leftover material.
  You can use a peeler to remove any film.
- Take the following precautions to maintain print quality and prevent further failures:
  - Replace the Tray Sheet—If residue persists, or the tray sheet is worn out, replace it to ensure optimal printing conditions.

  - Filter the Resin—Use a mesh filter to drain and remove any cured particle contamination from the resin tray, preventing defects in subsequent prints.
- Clean the Build Platform and Resin Tray—To eliminate any residual cured material that could interfere with adhesion or cause inconsistencies, thoroughly wipe down the build platform and resin tray, if needed.
- Check for Optical Clarity—Ensure the light source and optical path are free from dust or residue that could impact cure uniformity.
  Pay special attention to the glass and the bottom of the tray sheet, as any contamination here directly affects light transmission to the resin.
- Verify Printer Calibration—If failures persist, check that the projector irradiance and printer settings such as exposure time, intensity, and layer adhesion are within specifications to rule out process-related issues.

## Troubleshooting

- If a print failure occurs, and residue remains in the resin tray, it may lead to repeated failures in subsequent prints.
  To prevent this issue, we highly recommend using a regular mesh filter to filter the resin, and replace the tray sheet, if necessary.

Figure 4: Mesh filter to filter resin

- If a print failure occurs and the parts stick to the tray sheet instead of the build platform, increase *Exposure Duration,* adjusting the build platform adhesion settings, and then reprint the part, as described in Print Setup.



For more information about Stratasys systems, materials, and applications, visit www.stratasys.com.