POLYPROPYLENE FILAMENT

Printing Guide

Introduction

Braskem's polypropylene (PP) filament is specifically engineered for optimal performance in Additive Manufacturing. Polypropylene's mechanical properties, chemical resistance, low density, and ability to form living hinge features make it an ideal material for use in the Additive Manufacturing industry. This processing guide intends to provide you with the information you need in order to get the most from your Braskem PP filament. This guide has you covered, from preparing the build plate to slicing conditions and part design considerations.

Bed Adhesion Methods

Much like with other filaments, proper bed adhesion and bed levelling are key when printing with PP. Below are two methods that will ensure your success with your PP filament by providing the optimal resistance to warpage.

The methods described below are suitable for most geometries. However, warpage prone geometries containing sharp corners, large dimensions, or thick cross-sections may have varying results. **It is important to note that the success of any bed adhesion solution is heavily dependent on the levelling of the build plate and deposition of the first layer of filament**. Before using any bed adhesion product, refer to your printer's operating manual to ensure the build plate is precisely levelled. This will help guarantee proper deposition of the first layer against the bed.

In addition to the methods described below, there are various products which may work as a bed adhesive solution however, the solutions in this guide are recommendations to provide the most consistent and convenient solutions.

IMPORTANT: Traditional bed adhesion methods used in PLA and ABS printing such as painters tape, hair spray, and craft glue sticks will NOT provide proper bed adhesion with PP filaments. Parts will begin to warp and quickly detach from the bed. PP filaments have a unique material composition that is not compatible with these methods

PP Bed Adhesion Solution Stick

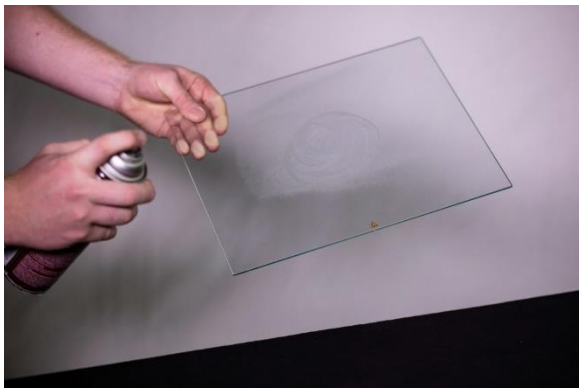
The preferred bed adhesion method is to use an adhesion solution stick specifically designed for PP 3D printing applications. Polypropylene adhesion solution sticks (sometimes referred to as adhesion glue sticks) are readily available online for purchase, provide great bed adhesion, and are water soluble making them easy to clean up. This method works best with glass surface build plates.

- First ensure the printer is cool to the touch and thoroughly clean the surface of the build plate to remove any potential contamination.
- Level your bed prior to solution stick application. It is critical the print bed is level to ensure proper deposition of the first layer of PP, enabling adequate adhesion to the bed.
- Remove the cap and apply a thin uniform coat of adhesive to the build plate using the foam or roll-on applicator using a circular motion, overlapping layers until the print area is covered. You will be able to see a solution form a haze over the glass.
- Adjust the printing conditions to ensure the printer has a heated build plate of 60-80°C per manufacturer's guidance. The actual temperature of the build plate may be different between machines; therefore, it is possible temperature adjustments outside of this range will be required.
- Once the print is completed and the build plate is cooled to room temperature, the part may be carefully removed from the build plate using gloves and a putty knife. For parts that are difficult to remove, heat the build plate to 80°C to help release the glue.
- It is possible to re-apply the adhesive following part removal for subsequent builds, but adhesion will be negatively affected by each re-application. The build plate can be cleaned using warm soapy water with an abrasive sponge. Do not clean the build plate while it is inside/attached to the printer.

IMPORTANT: Polypropylene bed adhesion solution stick works best for printing times <24 hours. When printing times are >24 hours, parts may begin to warp as a result of the extended exposure to the hot build plate. For parts which require print times of >24 hours, it is best to set the build plate to 80°C for the initial layer and 23°C for the remaining build time.

Spray Adhesive

Spray adhesives also work well with PP filaments, but it is recommended to only use this method for removable build plates. It is not recommended to apply spray adhesive directly to fixed build plates. Overspray can damage the other components of the printer and clean-up can be challenging.



- First ensure the printer is cool to the touch and thoroughly clean the surface of the build plate to remove any potential contamination.
- Level your bed prior to spray application. It is critical the print bed is level to ensure proper deposition of the first layer of PP, enabling adequate adhesion to the bed.
- Remove the build plate to eliminate any potential overspray on nearby rods, belts, or electronics. If your build plate cannot be removed, place a glass or mirror surface on top of the build plate and secure using binder clips (bed will need to be re-levelled to the new surface height). The adhesive spray works best on glass surfaces; other build plate materials may have non-stick properties that reduce the effectiveness of the adhesive spray.
- In a well-ventilated area, apply a brief spray of adhesive (2-4 seconds) that evenly coats the room temperature build plate. The build plate should appear to have a thin small bumpy texture. Spraying too much adhesive may result in the formation of a thick film and reduce the performance of the adhesive.
- Allow the adhesive to dry in a well-ventilated area for 1-2 minutes before heating up the nozzle.
- The bed is now prepared for printing. Optimal bed adhesion for spray adhesives occurs when bed heating is turned off or set to 20-30 °C.
- Once the print is completed, the part may be carefully removed from the build plate using gloves and a putty knife.
- Another brief coating of spray adhesive can be applied to the print area by following steps 4 & 5. It is possible to re-apply the adhesive following part removal for subsequent builds, but adhesion will be negatively affected by each re-application.
- The build plate can be cleaned using adhesive removers followed by a wash in warm soapy water using an abrasive sponge. Do not clean the build plate while it is inside/attached to the printer.

Slicing and Design Guide

Your slicing and design settings are another important component for ensuring the quality of your final printed part. Your Braskem filament comes with a product data sheet with some basic recommended printing conditions to help get you started. These are found in a table that looks like the one below. However, there are some additional settings that may need additional review such as first layer conditions, brim settings, raft settings, print speed, and support structures. Prior to adjusting any of these settings, double-check that the build plate is level. Inconsistencies in the height of the bed relative to the nozzle can result in cascading effects that will negatively affect your final part.

Parameter	**Units**	**Range**
Extruder Temperature	°C	220 - 230
First Layer Extruder Temperature	°C	235
Bed Temperature - PP Adhesion Solution Sick	°C	60-80
Bed Temperature - Multi-purpose adhesive spray	°C	20-40
Printing Speed	mm/s	35 - 65
First Layer Print Speed	mm/s	20 - 40
Fan Speed	%	50 - 100
First Layer Fan Speed	%	0
Flow/Extrusion Multiplier	–	0.90 – 1.10
Overlap Percentage	%	20 – 40
Brim	Layers	≥5
Raft Air Gap	mm	0.1

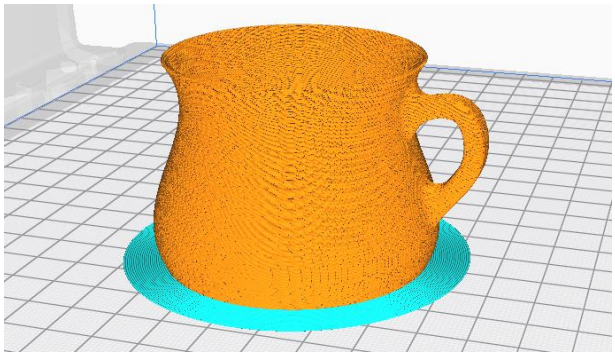
First Layer Conditions

The extrusion setting for the first layer of filament deposition is critical to ensure successful printing of your PP filament. The setting for the first layer will be different to ensure a strong interface with the bed and minimize warpage.

- Ensure the bed is levelled to ensure the first layer makes full contact with the bed.
- If the nozzle is too far from the bed at any spot along the first layer, this section will not adequately adhere to the bed and will begin to warp.
- Fan speed should be off or set to 0% for the first layer, this will allow the first layer more time to bond.
- Print speed should be slower (20-40 mm/s) for the first layer to ensure the nozzle deposits an even layer of polymer on the bed.
- Nozzle temperature should be set to 235°C for the first layer. Ensure the first layer is fully extruded and depositing consistently.
- Flow / Extrusion Multiplier should be set to 0.90 – 1.10.

Note: *If it seems that the first layer is under-extruding, increase the flow/extrusion multiplier to ensure sufficient deposition of material.*

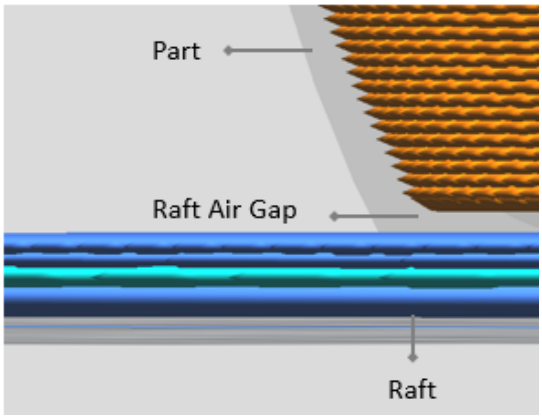
Brim Settings



Knowing how and when to utilize a raft and brim when printing with PP can be very important for printing various geometries using Braskem's PP filament. Using a brim is a quick way to enhance the adhesion of a part to the plate without significantly increasing the print time or material usage.

The brim width required will depend on the part geometry being printed. At least a 5-layer brim width is recommended to start, but this number should be increased for more complex or warpage-prone geometries.

Raft Settings



If a brim is not providing sufficient adhesion to the build plate for a given geometry, a raft can be used to provide increased adhesion between the part and the build plate. When using a raft, there are two main parameters to consider for adjustment: the raft air gap and extra raft margin.

Raft Air Gap: The most important parameter is the raft air gap. The raft air gap is the spacing between the topmost layer of the raft and the bottom most layer of the part. The ideal set point for the raft air gap is 0.1mm (this value may vary slightly up or down based on your printer's make and model).

Note: *If the raft air gap value is set too low, the part will fuse directly to the raft making the removal of the raft very challenging. If the raft air gap value is set too high, the part will not sufficiently adhere to the raft, and the part will warp or not adhere to the build plate at all.*

Raft Margin: Another important raft parameter is the extra raft margin, which is the distance the raft extends past the part itself. It is recommended to start with an extra raft margin of 10mm (this may need to be increased for more complex or warpage-prone geometries).

First layer print speed can also significantly influence the bed adhesion performance, please see the Print Speed section for more information.

Note: *It is important to note that bed adhesion is highly dependent on bed levelness, ensure the nozzle is level at all locations across the bed for optimal printing results. Also, please follow the bed adhesion guide in order to prevent warpage.*

Print Speed

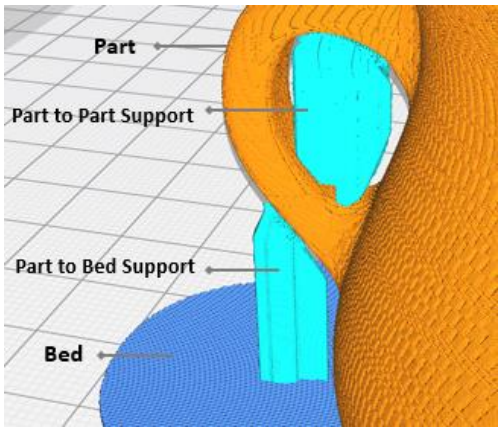
Print speed often determines both the velocity of the print head as well as the rate that material is extruded from the nozzle. This setting can vary quite significantly depending on the type of geometry being printed. The first layer print speed significantly influences the bed adhesion performance and should be slow to ensure intimate and precise contact with the build plate.

- The recommended print speed will depend on the geometry of the part
- It is recommended the first layer print speeds should be adjusted to 20-40 mm/s.
- For high-resolution prints which require mechanical strength, speeds less than 20 mm/s are recommended
- For high-speed printing, the print speed of the first layer can be increased up to 40-65 mm/s.
- For subsequent layers print speeds may be increased up to 35-65 mm/s

Note: *If your machine seems to be under or over extruding at any of these conditions, try adjusting the flow/extrusion multiplier parameter to account for these variations.*

Support Structures

Support structures are key to successfully printing complex geometries containing regions with direct overhangs or steep angles. These regions will require proper support in order to achieve the best surface finish and performance.



There are two types of support structures in the world of fused filament fabrication — structures that adhere between the build plate and the part, and structures that adhere between different levels of the part. For both types of supports, it is recommended to use only PP as the support material.

Similar to the raft parameters, support structures have a set air gap at the top and bottom of the support to enable easy support removal. This gap is typically set to increments of the layer thickness. Therefore, it is suggested to set the air gap to 1 layer for both the top and bottom of the support material. If the layer thickness is large (> 0.25 mm) and support structures are failing while using a 1-layer air gap, consider using an air gap of 0. This may require additional post-processing to remove the support structures but it will ensure adequate support of the geometry.

Note: *Other common support materials such as water-soluble supports, PLA, ABS, or PETG will not adhere to the PP. This lack of adhesion will most likely result in failure, especially for part-to-part support structures.*

Part Design for Warpage Reduction

In order to produce the highest quality parts with the least amount of warpage, it is important to understand what causes part warpage. There are several situations to be aware of when trying to minimize part warpage.

- **Thick cross sections** are more susceptible to warpage. This susceptibility is caused by uneven cooling between the exterior and interior of the part. Warpage associated with thick cross-sections can minimized by adjusting the design or reducing the infill density of the part.

- **Sharp corners** are often prone to warpage as they are stress concentration points. Warpage in these areas can be reduced by rounding corners or sharp edges.

- **Unsupported overhangs** are also very prone to warpage as they have a minimal amount of interaction with the previous layer. To reduce the impact of warpage on steep overhangs, it is recommended to add support structures, alter the design to reduce the overhang angle, or rotate the part in the slicing software to artificially reduce the overhang angle.

- **Thin walls** printed directly on the build plate can often lead to warpage as these features have minimal surface area in contact with the build plate. The warpage effect from this feature is significantly increased the higher the wall becomes or if the wall contains sharp corners (see “sharp corners”). Thin wall structures printed in circular patterns, similar to a vase, have better warpage resistance. Using a brim can help increase the contact surface area and prevent warpage.

Printing with Polypropylene Filaments FAQs

Please remember fused filament fabrication (FFF) machines can vary significantly between brands and models. Always refer to your printer's operator manual to ensure safe operation of your printer.

Why is my PP filament not adhering to the print bed?

Ensure your first print layer is set up for success.

- Check that you print bed is level, PP filaments have a low tolerance for un-level print beds. Please refer to your printer's operating manual for how to properly level your bed.
- Use a bed adhesion specifically designed for PP filaments. We recommend an adhesion glue stick specifically designed for 3D printing of PP or an adhesive spray. (NOTE: Painters tape and/or hairspray will not work for PP filaments)
- Be sure to adjust the first layer height to guarantee some pressure of the nozzle against the bed during material deposition
- Use a slower printing speed on the first layer (20-40 mm/sec);
- For complex parts try adding extra raft/brim layers to improve bed adhesion

---

Why is the material flow out of the nozzle slow or restricted?

Slow material flow is usually caused by an obstruction or an incorrect nozzle temperature.

- Ensure you have set the nozzle to be between 220-230°C (235°C for the first layer). NOTE: Always refer to your printer operating manual for maximum allowable nozzle temperature.
- Ensure the nozzle is not blocked by materials from prior prints. Refer to your printer's manual to determine proper way to clear blockages.
- Ensure the feeding system is properly driving filament into the hot end.
    - Overtightening the gear feeding system can result in the filament jamming or grinding out.
    - Under tightening the gear feed system can result in gear slippage which will underfeed polymer to the nozzle and reduce flow.
- Check your flow/extrusion multiplier to make sure it is set correctly and/or calibrated. It should be set to 0.90-1.10

Why are the parts warping during printing?	Warpage is typically caused by issues with improper bed adhesion (see FAQ regarding bed adhesion). If the bed has proper adhesion and levelling and warpage persists, fixes include: • Ensure the cooling fan is off for the first layer and set to 50-100% for the following layers. • Reduce the infill % or # of outline layers to reduce thermal accumulation within the part. • Remove sharp edges from the design to reduce stress build-up within the part. • Ensure steep overhangs are properly supported Polypropylene bed adhesion solution stick works best for printing times <24 hours. When printing times are >24 hours, parts may begin to warp as a result of the extended exposure to hot build plate. For parts which require print times of >24 hours, it is best to utilize the spray adhesive method.
Why is my part not sticking to the raft?	When this occurs please check the raft air gap. The raft air gap is the space between the topmost layer of the raft and the bottom most layer of the part. This needs to be set correctly to ensure proper adhesion between the part and the raft.
Why is my part sticking too much to the raft?	The raft air gap should be set to 0.1mm but further adjustment may be required depending on the printer used.
Why is my print warping during very long build times (>24 hours)?	Thermal exposure during long build times can cause warpage to occur in part. When utilizing a PP Bed Adhesion Solution Stick for long build times, it is recommended to set the build plate temperature to 80°C for the initial layer and 23°C for the remainder of the build.
Why are my part warping when printing multiple parts on the same build plate?	Ensure the build plate is level and clean at all points on the build surface. Minor variations in levelness or contamination on the surface can lead to warpage occurring in your parts.