ABS/ASA Usage Guide

1.Important Reminder

1.1. Ventilation & Safety

ABS and ASA filaments release irritating fumes and potentially harmful volatile organic compounds (VOCs) during printing. Ensure the printer operates in a well-ventilated area throughout the entire process. Avoid printing for extended periods in living spaces such as bedrooms or living rooms, and do not operate the printer in enclosed spaces.

2.Filament Overview

2.1. Filament Introduction

ABS and ASA are both amorphous thermoplastics known for their toughness, impact resistance, and heat resistance. They are widely used in FDM 3D printing to produce functional parts. While the two materials share many similar properties, each has its own strengths, and their printing parameters and processing requirements differ slightly. Below is a detailed guide.

Filament	Description	Applications	Print Difficulty
ABS	High toughness and impact resistance with moderate heat resistance	Functional parts	Medium (enclosure required)
ABS-Matte	High toughness and impact resistance, matte finish	Functional parts	Medium (enclosure required)
ABS-PRO	High toughness and impact resistance with better dimensional stability than standard ABS; crack-resistant	Functional parts	Medium (enclosure required)
HS-ABS+	High toughness and impact resistance with better dimensional stability than standard ABS; crack-resistant	Functional parts	Medium (enclosure required)

F.R.ABS	High toughness and impact resistance with good flame-retardant properties	Home appliance housings, lighting fixtures, and other flame-retardant applications	Medium (enclosure required)
ESD-ABS	Good heat resistance, high toughness, high impact resistance, good antistatic properties	Electronics enclosures, jigs, fixtures, and manufacturing tools	Medium (enclosure required)
ASA	High toughness, impact resistance, and weather resistance, with moderate heat resistance and antistatic properties	Functional parts	Medium (enclosure required)

2.2. Filament Properties

	Strength (Flexural Strength - XY)	Toughness (Notched Impact Strength - XY)	Heat Resistance (HDT @ 0.455 MPa)
ABS	65~70 Mpa	7~10.5 KJ/m²	88 °C
ABS-Matte	73.5~74.5 Mpa	25.5~29.8 KJ/m²	88 °C
ABS-PRO	65~67 Mpa	12~13.5 KJ/m²	85 °C
HS-ABS+	65~67 Mpa	12~13.5 KJ/m²	85 °C
F.R.ABS	55~60 Mpa	10~12 KJ/m²	86 °C
ESD-ABS	60~61 Mpa	6~7 KJ/m²	98 °C
ASA	75~79 Mpa	19~20 KJ/m²	88 °C

2.3. Hardware Compatibility

Filament	Printer Compatibility	Nozzle Compatibility	Build Plate Compatibility	Accessory Compatibility
ABS/ASA	Adventurer 5M (enclosure kit required) Adventurer 5M Pro AD5X (enclosure kit required) Guider 3 Ultra	Compatible with all nozzles	Glue required; not compatible with Cool Plate	Compatible with IFS

3.Preparation Before Printing

3.1. Filament Drying

ABS and ASA are hygroscopic. Moisture absorption can lead to bubbling, stringing, and poor layer adhesion during printing. Drying before printing is essential.

Filament	Air Drying Oven	Heated Bed
ABS	75-85 ℃，8-12h 75-85°C, 8-12h	90-100 ℃，12h 90-100°C, 12h
ASA	75-85 ℃，8-12h 75-85°C, 8-12h	90-100 ℃，12h 90-100°C, 12h

When drying on the heated bed, flip the spool every 6 hours and cover the filament with its packaging box or a PC box to ensure even heating.

3.2. Nozzle

Clean the nozzle thoroughly and ensure it is free of debris.
Note: If you switch to a lower-temperature filament like PLA after printing with ABS/ASA, first heat the nozzle to the ABS/ASA printing temperature. Then manually tap the extrude button to load the filament. Once the filament flows smoothly from the nozzle, lower the temperature to the PLA printing temperature before tapping the extrude button again.

3.3. Build Plate

1. Clean the build plate to ensure the surface is free of debris or contaminants.
2. Apply glue: We recommend applying official glue to improve bed adhesion and increase print success rates.

3.4. Enclosure

1. ABS and ASA are prone to warping due to their high shrinkage rate and uneven temperature distribution during printing. Different areas of the model cool and shrink at different rates, generating internal stress that pulls corners off the build plate. Therefore, Adventurer 5M and AD5X should be used with an enclosure kit to prevent warping and cracking caused by temperature fluctuations.

2. Increasing the chamber temperature appropriately can effectively reduce internal stress caused by material shrinkage during printing, thereby improving the success rate and print quality. Increasing the bed temperature indirectly raises the chamber temperature. You can preheat the bed to its maximum temperature for 15 minutes before starting the print.

4.Core Print Settings

4.1. Temperature Settings

Filament	Nozzle Temperature	Bed Temperature	Chamber Temperature
ABS	250-270°C	90-100°C	40-60°C
ASA	250-270°C	90-100°C	40-60°C

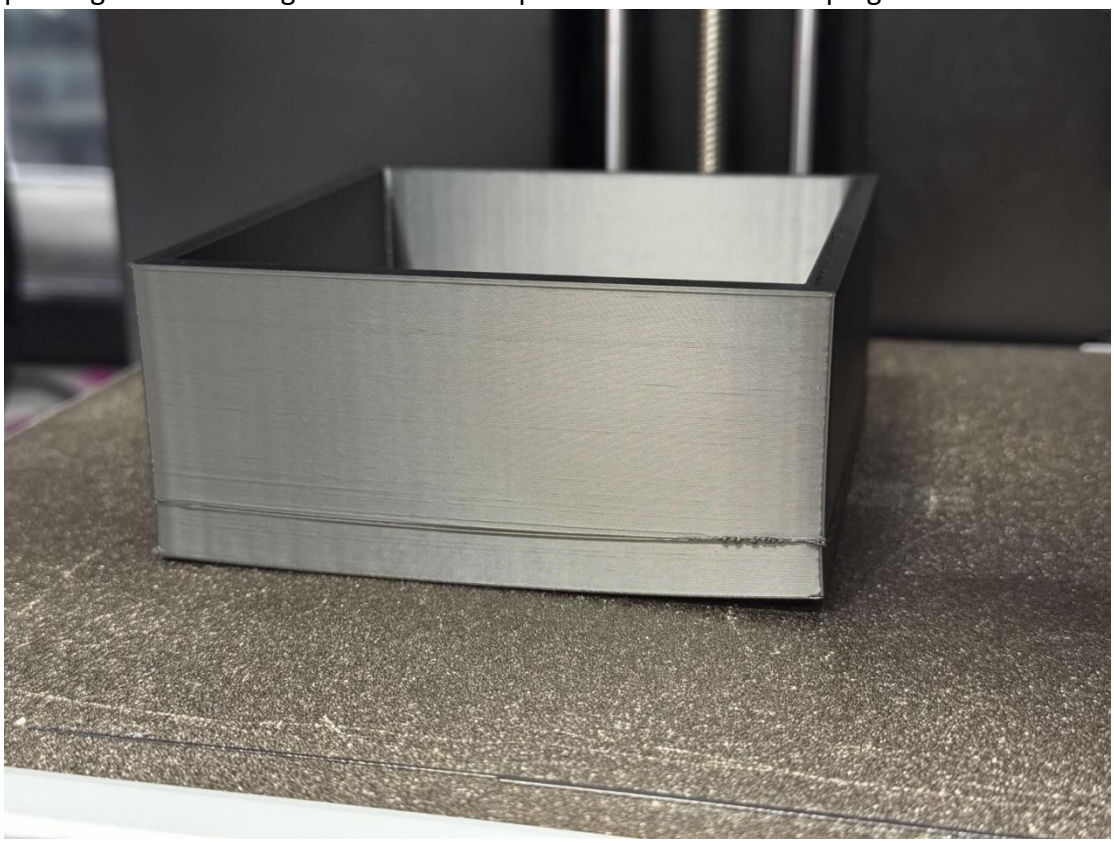
4.2. Cooling & Infill Settings

Filament	Cooling Fan	Infill	Wall Loops
ABS	0-20%	15% for non-structural parts, ≤50% recommended for structural parts	2 for non-structural parts, ≤6 recommended for structural parts
ASA	0-20%	15% for non-structural parts, ≤50% recommended for structural parts	2 for non-structural parts, ≤6 recommended for structural parts

5.Common Printing Issues & Solutions

5.1. Model Warping

Compared to PLA, ABS and ASA are more prone to shrinkage and warping during printing. The following methods can help reduce the risk of warping.



5.1.1. Cause Analysis

The primary cause is high material shrinkage combined with uneven temperature distribution during printing.

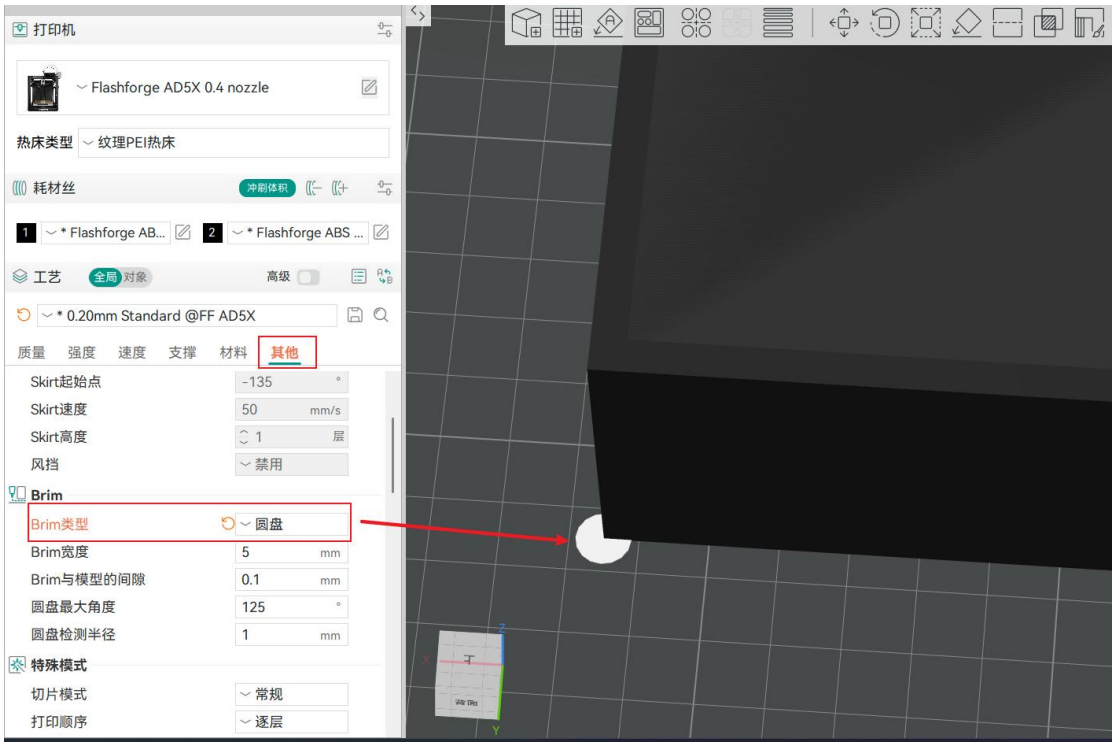
1. Printer not enclosed, leading to significant temperature fluctuations
2. Bed temperature too low
3. Cooling fan speed too high
4. Improper infill and wall loops settings

5.1.2.Solutions

1. Use an enclosure kit to reduce temperature fluctuations inside the chamber.
2. Preheat the bed to its maximum temperature for 15 minutes before starting the print to raise the chamber temperature.

3. Lower the fan speed and keep it within 0-20%.
4. Apply glue to the build plate and enable leveling before printing.
5. Optimize slicing settings:

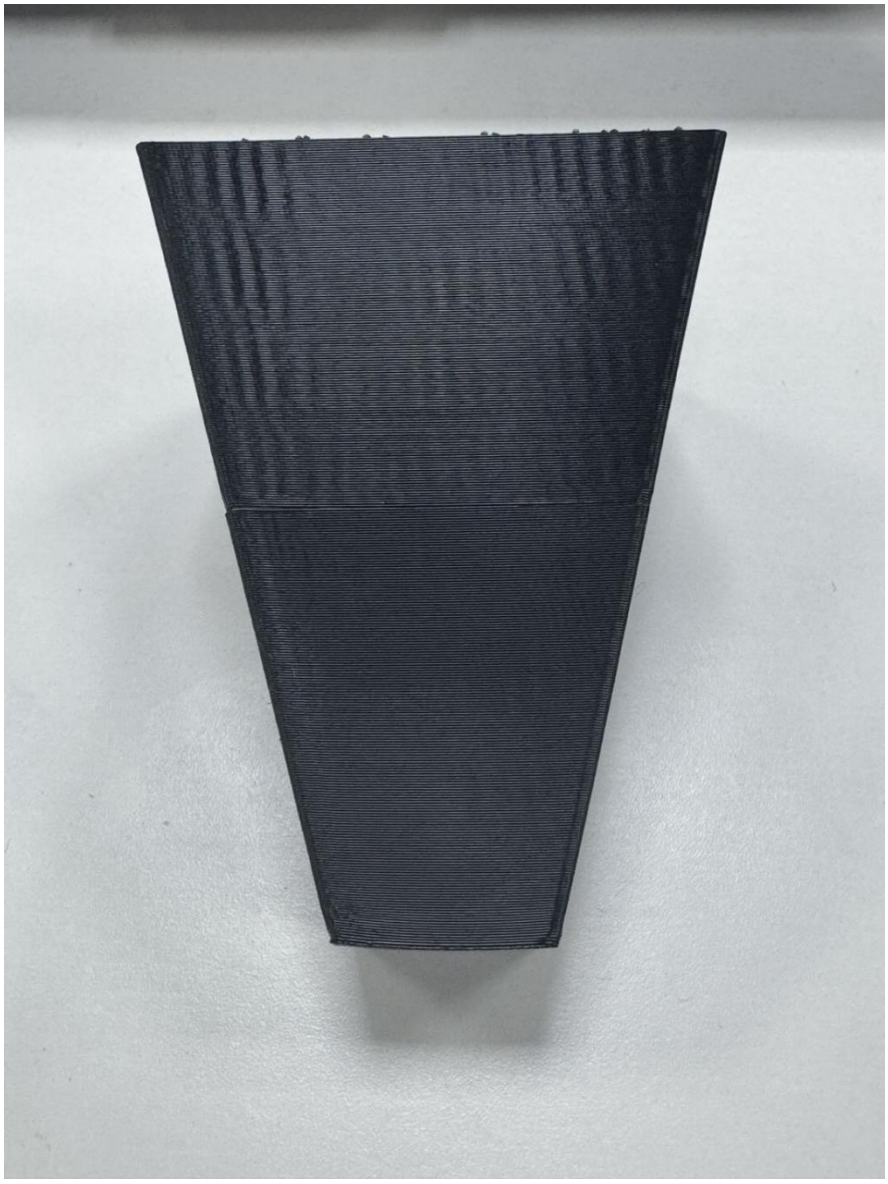
➢ Enable a brim: Increase the contact area between the model and the build plate.



➢ Keep infill below 50% to reduce material shrinkage.

➢ For non-structural parts with low strength requirements, **15% infill + 2 walls** is generally sufficient.

5.2. Poor Layer Adhesion/Interlayer Cracking



5.2.1. Cause Analysis

The root cause is that the internal stress from cooling during printing exceeds the interlayer adhesion strength, leading to visible cracks or layer separation. This issue is common when printing high-temperature filaments such as ABS and ASA.
The problem can generally be traced to one or more of the following:

1. **Under-extrusion**
   - Filament not fully melted, resulting in poor flow
   - Nozzle clogging causing inconsistent extrusion
2. **Adhesion or structural issues**
   - Insufficient interlayer adhesion

- Local areas of the model too thin

3. Excessive cooling

- Cooling fan speed too high
- Printer not enclosed, leading to significant temperature fluctuations

5.2.2. Solutions

1. Under-extrusion

- Run a temperature tower test to determine the optimal printing temperature for the filament, then increase it by 5-15°C.
- Reduce the print speed appropriately.
- Perform a cold pull, clean the nozzle with an unclogging pin tool while in a heated state, or replace the nozzle if needed.
  Cold pull: Heat the nozzle to the printing temperature, manually feed a short length of filament, then let it cool to approximately 90°C (PLA) or 150°C (PETG). Quickly and steadily pull out the filament to remove carbon buildup inside the nozzle.

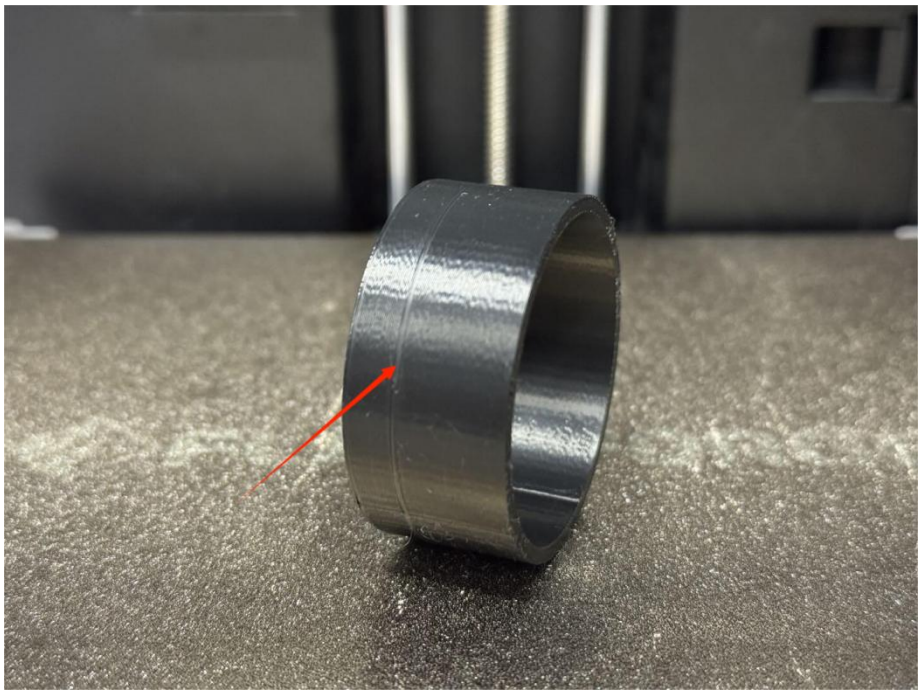
2. Adhesion or structural issues

- Increase wall loops or infill density appropriately to enhance overall strength and interlayer bonding.

3. Excessive cooling

- Lower the fan speed appropriately.
- Use an enclosure kit, close the front door, install the top cover, and increase the bed temperature to maintain a stable chamber temperature.

5.3. Horizontal Lines Around the Model (Cooling Artifacts)



5.3.1. Cause Analysis

Large time differences between layers cause uneven shrinkage. These lines often appear at the junction between the large base and thin walls of box-shaped models, or around the midsection of small boat models.

5.3.2. Solutions

- Modify the model in modeling software (such as SolidWorks or Fusion 360) by adding an inner fillet.

5.4. Surface Bubbles & Stringing

5.4.1. Cause Analysis

- Moisture absorption (wet filament)
- Improper retraction settings

5.4.2. Solutions

- Dry the filament using a filament dryer or heated bed.
- Reduce the extruder temperature slightly and increase the retraction length appropriately.

6.Post-Processing

6.1. Support Removal & Sanding

- PVA supports can be dissolved directly in warm water.
- Same-material supports can be removed with a craft knife or pliers. Sand away any remaining marks progressively using sandpaper (120 grit → 400 grit → 800 grit) for a smooth finish.

6.2. Mechanical Polishing

- After sanding, apply polishing compound to enhance surface gloss.

7.Filament Storage After Printing

7.1. Sealed Storage

- Store the filament in a sealed bag with desiccant at room temperature.

Note: It is recommended to print immediately after drying to prevent the filament from absorbing moisture again.