

Exit Ticket 1

1. Input

is it a problem when i apply a 10mm radius on the bottom edge of my 3d printing part?

Output

Short answer: it can be.

A big bottom-edge radius (like 10 mm) means your part touches the build plate with a very small contact area at first. On FDM printers this often leads to poor adhesion, warping, or the part popping loose mid-print—especially with ABS/nylon. It can also exaggerate “elephant’s foot.”

Better approaches

- **Add a small flat or chamfer:** Leave a 0.8–1.5 mm flat around the base, *or* a 0.5–1 mm 45° chamfer, then start your 10 mm radius above that. You keep the look without sacrificing adhesion.
- **If you must keep the radius to the bed:**
 - Use a **brim** (5–10 mm), or “mouse ears” tabs on corners.
 - Slow and thicken the first layer; reduce early cooling; ensure bed temp/adhesive is dialed in.
 - Consider printing on a **different face** so the radius isn’t the first layer.

Material notes

- **Most risky:** ABS, nylon, PC (strong warping).
- **Moderate:** PETG (can also string and lift).
- **Most forgiving:** PLA (still benefits from a small flat/chamfer).

TL;DR

A 10 mm bottom fillet is usually a bed-adhesion liability on FDM. Give the part a small “landing” (flat or chamfer) and start the large radius slightly above the build plate. For resin (SLA) prints on supports, it’s less of an issue.

2. What I like about this prompt is, that it addresses a real-world problem and not something purely theoretical. Additionally, I like to prompt about 3D-printing due to my background as a design engineer.

3. The consequence of trusting the output blindly is only a design or printing failure caused by a hallucination or oversimplification, which is not that big of a deal. Blindly following the output could lead to unnecessary waste in money, time and/or material.
4. I saw that the output said a lot about the material type, which was not important to me, because I usually only 3d-print with PLA. So, I could've specified the material type within the prompt.

5. A/B Test Logbook

I tested my prompts with the model "GPT-4 Turbo".

Goal

Identify risks of applying a 10 mm radius on the bottom edge of a 3D printed part

Prompt A → Output

is it a problem when i apply a 10mm radius on the bottom edge of my 3d printing part?

Short answer: it can be.

A big bottom-edge radius (like 10 mm) means your part touches the build plate with a very small contact area at first. On FDM printers this often leads to poor adhesion, warping, or the part popping loose mid-print—especially with ABS/nylon. It can also exaggerate "elephant's foot."

Better approaches

- **Add a small flat or chamfer:** Leave a 0.8–1.5 mm flat around the base, *or* a 0.5–1 mm 45° chamfer, then start your 10 mm radius above that. You keep the look without sacrificing adhesion.
- **If you must keep the radius to the bed:**
 - Use a **brim** (5–10 mm), or "mouse ears" tabs on corners.
 - Slow and thicken the first layer; reduce early cooling; ensure bed temp/adhesive is dialed in.
 - Consider printing on a **different face** so the radius isn't the first layer.

Material notes

- **Most risky:** ABS, nylon, PC (strong warping).
- **Moderate:** PETG (can also string and lift).
- **Most forgiving:** PLA (still benefits from a small flat/chamfer).

TL;DR

A 10 mm bottom fillet is usually a bed-adhesion liability on FDM. Give the part a small "landing" (flat or chamfer) and start the large radius slightly above the build plate. For resin (SLA) prints on supports, it's less of an issue.

Prompt B → Output

You are an expert in 3D printing and design for manufacturability. Evaluate the potential risks of applying a large bottom-edge radius to a 3D-printed part. The part will be printed on an FDM printer using PLA filament, oriented so that the bottom edge (with the radius) is in direct contact with the build plate. Provide a clear, concise explanation of at least one possible issue, with a short suggestion to mitigate it. What problems could arise from applying a 10mm radius to the bottom edge of the part in this setup?

Short answer: a 10 mm bottom-edge **radius** that results in a *curved* contact with the build plate causes **poor/unstable first-layer behaviour** and a handful of related print problems. Below are the main risks and short mitigations.

Key problems & fixes

1) Poor bed contact → weak/unstable first layer (most important)

Why it happens: a large fillet makes the very bottom of the part **tangent** to the bed at a small contact patch rather than a broad flat area. The nozzle will try to lay the first layer onto a curved surface so either only a tiny chord touches (poor adhesion) or the filament is squished unevenly. Result: part can lift, shift, or fail early in the print.

Mitigation: give the print a small flat on the bottom (1–3 mm) or re-orient the part so a flat face contacts the bed. If you must print the curve down, use a wide **brim/raft** (20–30 mm brim or raft) and stronger bed adhesion (glue stick/PEI + correct bed temp).

2) First-layer thickness / Z-offset and nozzle collision issues

Why: slicers assume a flat bottom when computing toolpaths and initial Z offset. With a curved surface the nozzle can either be too close at the tangent (dragging/scraping) or too far elsewhere (poor extrusion). This causes inconsistent extrusion, filament blobs, or surface scarring.

Mitigation: avoid curved-down orientation; if unavoidable, raise initial layer height slightly and increase first-layer flow/line width, test and tune Z-offset carefully, or use a raft so the nozzle prints a flat sacrificial surface first.

3) Slicer artifacts / thin first-layer extrusion & gaps

Why: near the tangent the slicer may produce very narrow toolpath widths or tiny contact islands (the curved slice of the fillet) that are too small for your nozzle, producing gaps or weak infill at the base.

Mitigation: ensure features at the base are larger than the nozzle width (avoid features thinner than $\sim 0.6 \times$ nozzle diameter), increase first-layer line width, or use a raft/brim.

4) Dimensional inaccuracy and cosmetic defects at very low Z heights

Why: the squish and cooling behavior of the first few layers on a curve can shift geometry relative to the designed radius; edges can get flattened or distorted.

Mitigation: move the fillet away from the base (make the fillet start a few mm above the bed) or print the part on its side so the radius is not the initial contact surface.

5) Risk of part rocking / vibration

Why: a curved base with small contact zones can let the part rock as layers build, causing ringing or layer misalignment.

Mitigation: same as #1 — flatten the bottom or add broad temporary adhesion (brim/raft) and slow down initial layer speeds.

Practical recommendation (concise)

If the design intent allows: **add a 1–3 mm flat** where the part meets the bed (or re-orient so a flat face sits on the plate). If you cannot change orientation or geometry, use a **raft + wide brim**, increase first-layer extrusion/line width, and carefully tune Z-offset and initial layer speed. Those changes recover reliability without sacrificing the large fillet above the contact region.

If you want, tell me your nozzle diameter, layer height and intended first-layer settings and I'll give exact numbers for brim width, raft settings, and suggested flat thickness.

Which won?

The R-T-C-F-Q prompt won.

Why?

The answer from the R-T-C-F-Q prompt is way more detailed and doesn't address unnecessary things such as material types or printing techniques.

Next tweaks

Add more constraints in R-T-C-F-Q (e.g. print speed, bed surface, ...) to see if output becomes even more actionable.