



LOKSTER | SPACE

A HOME FOR ALL MY PROJECTS



AdChoices

[Repair Estimate](#)

[Printer Setup](#)

How to make Cura accurately estimate the print time

lokster 2018-03-18



For many users this is all too well-known "issue": Cura says your print will take 15 hours, you hit "Print!", and after 20 hours, it's still f***ing printing...

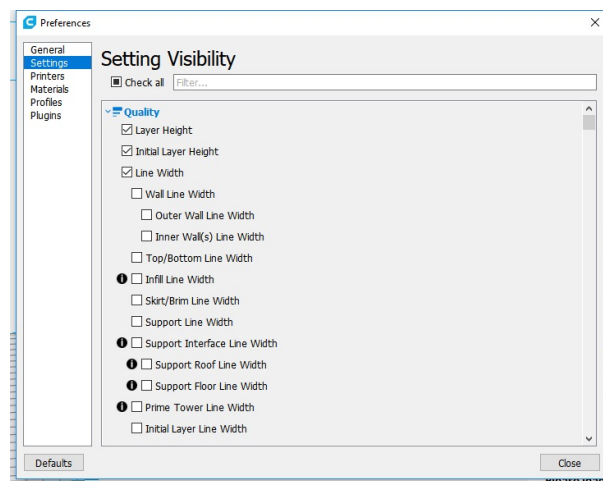
News flash: this does not affect only Cura. It affects every slicer to a different extent – depending on how well it's default settings match your printer's settings.

So... how to fix it?

It's quite easy actually. There are a couple settings you have to set correctly, and the time estimate will become very accurate. Like, just minutes difference in a multi-hour print!

OK, let's do it!

First – start Cura, click on the Preferences -> Configure Cura menu, and select the "Settings" section:



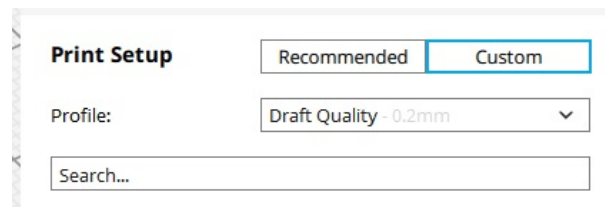
Note: stop using obsolete Cura versions! The "but 15.04 works fine" excuse is equal to "I'm too lazy to learn something new".

Now after I vented the pressure ^^, find the following settings, and make sure all of them have check mark (so they are visible when you are in “Custom” print setup mode):

- Enable Acceleration Control
- Print Acceleration
- Travel Acceleration
- Enable Jerk Control
- Print Jerk
- Travel Jerk
- Enable Retraction
- Retraction Speed

Mark them all, and click “Close”.

Now, select “Custom” print setup:



Put a checkmark on “Enable Acceleration Control”, “Enable Jerk Control”, and “Enable Retraction”.

Now, there are a few questions you have to answer:

1. Do you already use acceleration control (e.g. you have it already enabled, and set your own Print / Travel Acceleration values)?

- **YES:** make SURE the print and travel acceleration is NOT higher than the maximum acceleration value, defined in your firmware, otherwise the printer will just use the maximal allowed acceleration, and the actual print time will be longer than the estimated.
In Marlin's Configuration.h, this parameter is called “DEFAULT_MAX_ACCELERATION”.
You can also view your current limits from the printer's menu: Control -> Motion -> Acceleration -> Amax X (and Amax Y).
- **NO:** Set the print and travel acceleration values IN CURA to match the values of your printer.
In Marlin's Configuration.h the parameters are called DEFAULT_ACCELERATION and DEFAULT_TRAVEL_ACCELERATION.
Match your slicer's print acceleration to the first, and travel acceleration to the latter. **ALWAYS enable acceleration control if you want accurate print time estimation.**

2. Do you use Jerk control?

- YES: nothing to do here. Just keep the setting enabled!

- **NO:** Enable it. Put a checkmark on “Enable Jerk Control”, and match the Print Jerk / Travel Jerk to the values used by your printer’s firmware.

In Marlin’s Configuration.h, the parameters are called DEFAULT_XJERK and DEFAULT_YJERK.

Note: In Marlin, the jerk is set “per axis”, while the Cura jerk settings set it for both axes simultaneously.

So, just take the DEFAULT_XJERK value, and set it as your Print Jerk and Travel Jerk value.

You can check your current jerk values from your printer’s menu Control -> Motion -> Jerk -> Vx-jerk/Vy-jerk

3. This is the most frequently overlooked trick: The **retraction speed**.

Your firmware has maximum retraction speed limit just like it has limits for the other speeds.

So, for example, if you are using Anet A8 printer with Marlin firmware, and you have set your acceleration speed in Cura to 70mm/s (like me), your print time estimate WILL be inaccurate, because the maximum retract speed, defined in Marlin’s Configuration.h is 50mm/s.

It’s defined in DEFAULT_MAX_FEEDRATE – the fourth value (the “E” max feedrate, as in “extruder maximum speed”).

Do you use retraction?

- **YES** (most likely): make sure your retraction speed is not higher than maximum retraction speed, defined in your firmware.
You can also increase the firmware limit for the retraction speed e.g. you can set it to 70mm/s on both places – Cura & Marlin.
Personally, I use 70mm/s retraction speed on both places.
- **NO:** I’m not sure who does not use retraction 😊 Anyways, you don’t have to do anything if you don’t.

And that’s all! These couple of tricks will make your estimated print time a lot more accurate.

Leave a comment...

0 Comments

Sort by **Newest**

Facebook Comments Plugin

Share this:



CATEGORIES

3D Printing

Previous article

Bad print quality with USB or Octoprint? The solution is here...
or not

LEAVE A REPLY

Comment

Name *

Email *

Website

POST COMMENT

Hosting Partner

