Ender 5-Plus Operation Manual

Ender 5-Plus printers can only run g-code generated in the <u>Creality Print</u> slicer. All other slicers will result in feeding issues!

1) **Generating g-code**
   a) **Take .stl file and import it into Creality Print. Navigate to Profile and select edit icon.**

Print Temp	Use temp printed on filament
Build Plate Temp	50°-60°C
Adhesion	Skirt is usually sufficient
Print Speed	50.0mm/s MAX
Initial Layer Speed	15.0mm/s
Layer Height	<u>Height (Quality/Speed)</u> .10mm (Best/Slow) .20mm (Better/Normal) .30mm (Good/Fast)
Support?	Avoid if possible
Infil	LESS IS MORE! 10%-30% works for most prints.

2) **Load print**
   a) **Slice and export the file onto the provided microSD cards and insert it into the printer. Navigate to print, then select the file and begin.**
   b) **Ensure build volume and nozzle are clear.**
   c) **The printer will take a few minutes to pre-heat, this is normal.**
   d) **Applying a thin layer of glue stick glue to build the plate can greatly improve build plate adhesion.**
3) **Start Print**
   a) **The Ender-5 Plus can be left to print without supervision, however, it is highly recommended to observe the first layer or two of each print. Prints are most likely to fail in the first or last few layers.**