Kevin's Adhesion – Method and Tips

After seeing an influx of people posting with adhesion issues, I thought I'd share what's working for me 100% of the time now… No, I didn't buy a build plate. It's blue painter's tape, but with a twist… I'm still new to 3D printing so please keep that in mind. You may not have the same success, but it's worth a try if nothing else has worked.

Like several others (not all), adhesion was my first hurdle to overcome. Technically bed-leveling could be considered the first one, but I'm not going to get much into leveling methodology in this document. I made sure my glass bed was as clean as possible, but the results were hit or miss (mostly miss). Then I tried the washable purple Elmer's glue stick and other glues, Aquanet and other hairsprays, blue painters tape and masking tape. I leveled and releveled, leveled and releveled, etc. I tried various settings like squishing, flow, no fan, etc. All great tips from this group, but nothing seemed to be working consistently.

I decided to play with the painter's tape method some and everything changed. I have absolutely no signs of my prints lifting now. As a matter of fact, my adhesion is too good at times. Most people have more luck with removal once the bed is cold, but my prints come off much more easily when it's still hot. I suspect that's because the glue from painter's tape is still soft and more forgiving. Therefore the tape lifts from the bed ever so slightly, allowing you to slide your tool under the model more easily. I've test printed various models, with various profiles, sizes and challenges. They all stick, period.

In short, I found that the key was scoring the tape with rubbing alcohol, bringing just enough glue from the tape through to the surface.

Another thing I've learned is that the bed springs need to be worked in a bit (repeated compression). They just seem to cause leveling issues until they've settled in.

First, a little info on my setup and supplies:

1) 2018 Gold Tevo Tornado, all stock (basically)

2) Silicone sock
https://www.amazon.com/dp/B07FJVYFZQ/ref=cm_sw_r_tw_dp_U_x_hHmwCbTGV54QY

3) IdeaMaker (See settings at the end of this document. They may not all be ideal, but good recommendations to start with)

4) Inland Premium 1.75mm Light Blue PLA+ printed at 205C/50C
https://www.microcenter.com/product/504216/Premium_175mm_Light_Blue_PLA_3D_Printer_Filament_-_1kg_Spool_(22_lbs)

5) eSun Multi-function 3D Printing Filament Storage Box with Silica Desiccant
https://www.microcenter.com/product/509681/Multi-function_3D_Printing_Filament_Storage_Box
and
https://www.amazon.com/dp/B00DYKTS9C/ref=cm_sw_r_tw_dp_U_x_dTmwCbYCZEVMV

6) 2.82 inch wide ScotchBlue Painter's Tape (3M) https://www.amazon.com/ScotchBlue-Painters-Multi-Use-2-83-Inch-60-Yard/dp/B001EJMS4M

7) 70% isopropyl alcohol

8) Paper towels (I use Bounty Basics)

9) Lint-free cotton cloth (like a T-shirt)

<u>I suggest the following preliminary measures:</u>

1) Check the bed heater – Mine was sagging/drooping from the glass, which was creating inconsistent bed temps. To address this, I heated the bed to about 70C, carefully but quickly (so it didn't cool too much) removed the glass bed from the printer, flipped it glass-side down on a flat and sturdy surface, balled my hands up and rolled the top/flat part of my fists into the foil backing. Careful not to press too hard, but enough pressure to press the pad back to the glass, holding for a couple seconds each time to help it set. I made kind of a grid-looking indentation into the foil, since I rotated it 90 degrees and kneaded the surface from the perpendicular direction as well. Then I let it cool upside down so the glue set, before I put the bed back on the printer… I haven't noticed any continued sagging, but if I do I'll be revisiting this and applying more glue.

2) Silicone sock – While I'm not 100% certain that this helped with adhesion, my amateur brain thinks it has some influence in regulating the heat better than the original garbage foam and tape. The added benefit is that I no longer have filament buildup around my nozzle. It just wipes away or falls off on its own. Installation was pretty straight forward. I raised my Z axis so that I could easily see under the hot end/fan housing (you could always remove the housing if you're modding this assembly anyways, but I didn't at the time). I carefully cut across the yellow thermal tape, releasing the white foam/fluff underneath. I used a pair of tweezers to guide the tape around/off the hot end and to remove the leftover foam/fluff. Blow out the remaining foam/fluff with a can of air, then proceed to applying the silicon sock. The one listed above fits, even though the nozzle cut-out is off-center from the actual nozzle. It only fits one way though. The side that reads "MK9" should face the front of the printer. Make sure the sock wraps completely around and over the top of the block, so it doesn't create a problem later. The sock actually folds around the top of the hot end, just enough to catch, but mine didn't want to at first. I guess it softened after a few prints because now it fits like a glove. Just keep an eye on it and adjust if needed.

3) Drying and heating the filament – You should keep your filament in an air-tight container with silica desiccant at all times. Moisture will ruin your prints. The eSun box listed above seems to make some difference, but I'm not completely sold on it. I keep whatever filament I'm currently using in it, including a couple packs of silica desiccant. I set the temp on the box to 50C, about 30 minutes before I print (longer if possible and manually rotate the spool to heat evenly).

<u>Now, to prepare the bed for adhesion:</u>

1) Move the Hot End (Z axis) – This can always be done with steps from the control box or computer/octoprint. If you move it manually, make sure you have either disabled steppers or turned the printer off. To move it manually, use your fingers to spin the coupler (around the threaded Z rod). Raise the Z axis so that the hot end carriage is up and out of your way.

2) Move the bed forward (Y axis) – Move the Y axis forward, so that you can completely access the bed's surface. Again, either manually with steppers turned off or from your computer/octoprint. If you're comfortable working under the frame, skip this step. I found it much easier to bring the bed all the way forward so that the frame wasn't obstructing access to the bed surface.

3) Apply blue painter's tape – Although it's been discussed extensively, I haven't really seen anyone mention what I've found makes the tape adhere to the print so well (outlined in the next steps). I always make sure the bed has cooled down before preparing the surface and make sure it's really clean before proceeding… I usually measure a piece of tape out, cut and apply it squarely on the bed, starting in the front of the bed. Make sure it's completely flat when you place the tape (no bubbles, wrinkles, folds or ripples). Try to minimize the amount of oils from your fingers. You'll get most oils up in the next step, but the less oils the better… I start in the front of the bed, with the first strip, then work my way back on the bed. My first strip is actually cut in half length-wise (flat edge facing back) and I've never printed on it, so I've never pulled it up and use it as my starting guide. It's followed by 3 of the 2.82 inch wide strips of tape. This is enough to cover my bed, between the bed screws and enough for most prints/models… When applying each strip, make sure you carefully line the tape edges up so that there is no overlap or gaps between the tape strips. Press firmly and evenly as you apply the tape (again with minimal touching/oils, no bubbles, wrinkles, folds or ripples). Now that the bed is covered, the next steps made all the difference for me…

4) Making the tape tacky – I've only read of one other person mentioning this, but for me everything changed for the better once I started doing this… The bed should be cooled off so the alcohol doesn't evaporate too quickly. Grab a paper towel, fold it up, apply 70% isopropyl alcohol (apply enough, but do not saturate). Wipe down the painter's tape (again, do not saturate the tape), swiping back and forth until you start to feel the painter's tape grabbing the paper towel. Try and make the friction consistent across the entire surface. Monitor the amount of alcohol you're applying by looking at the "wetness" before it evaporates. Uniformity will be your friend here. As it starts to get slightly tacky, you'll begin seeing bits of paper towel sticking (tiny pieces, if not it's probably too much alcohol). You want this paper towel-fuzz to be subtle and uniformed. If you're not getting any tackiness, apply more alcohol. If you're not getting enough paper towel-fuzz, use a fresh/dry paper towel. Lastly, take a lint-free piece of cotton cloth (I use a T shirt) and gently wipe away any loose paper towel-fuzz.

Additional tips for a good first layer(s):

1) Start by printing slowly – I started at 20-30%, but now I just kick it off at 100% and everything sticks perfectly.

2) Bed temp – The sweet spot will vary between filaments. I ran Inland's PLA at 50C for the first layer, then dropped it to 45 after because I was seeing some elephant foot and discoloration in the following few layers. I'm currently printing with CC3D's silk PLA and I have to keep it at 50C the whole time, with no warping, discoloring or lifting.

3) Cooling/Fan Speed settings – Using IdeaMaker, I run the first layer with 0% fan, then 25% for layer 2, then 75% for layer 3 and 100% for the rest. I could probably skip some of the interim fan speeds, but it's working well as is (see "Cooling" screenshot at the end of this document)

4) Controlled cooling/fan mods – I now have Bullseye installed to ensure that the print is cooling evenly and not just on one side of the model https://www.thingiverse.com/thing:2759439

5) Ambient temperature changes – I now use a custom enclosure so that my ambient temperature is consistence. Personally, I was fighting drafts in my home so I had to come up with something. I used this tutorial, but used the foil side in and made sure everything was as precise as I could make it https://youtu.be/FXMy5i389g0. You don't want your control box inside the enclosure, so you'll either have to cut out the side of the enclosure like they did, order a set of extension cables from Tarek or come up with something on your own.

Post-tape leveling tip:

Since I don't put tape all the way to the corners of the bed, I allow for a larger gap, when adjusting the tension springs. I use a .35mm thick card (like a playing/poker card) and adjust each corner until the nozzle starts to grab the card. I always start with Auto Home, then check each corner multiple times. As of late, I rarely have to adjust the tension (now that the springs have settled in). When the nozzle travels across the painter's tape I can see that the gap is less and this works perfectly for me.

I still feel like this method of adhesion is "cheating", but it helped me start printing and enjoying my printer. Now that I'm more comfortable with the printer, I'm going to start experimenting with printing directly on the bare glass.

My current IdeaMaker Settings:

I'm sure other members will disagree/comment on some of these. I welcome any feedback because I am still learning.

Advanced Settings

Layer | Extruder | Infill | Support | Platform Additions | Cooling | Temperature | Advanced | Ooze | Other | GCode

Primary Extruder

General

Extrusion Width: `0.40` mm

Retraction

☑ Enable Retraction

Retraction Speed: `63.0` mm/s

Retraction Material Amount: `8.0` mm

Minimal Travel of Retraction: `0.3` mm

Minimal Amount of Retraction: `0.02` mm

Extra Restart Amount: `0.00` mm

Restart Speed: `25.0` mm/s

Z Hop at Retraction: `0.000` mm

☐ Z Hop Only at Solid Parts

Coasting

Coasting Distance: `0.50` mm

☐ Apply Coasting on Grid Infill and Lines Solid Fill

Wipe

Outer Shell Wipe Distance: `1.00` mm

Outer Shell Wipe Speed: `60.0` mm/s

Advanced Settings

Layer | Extruder | Infill | Support | Platform Additions | Cooling | Temperature | Advanced | Ooze | Other | GCode

Infill

Infill Extruder: `Primary Extruder`

Infill Density: `25` %

Infill Speed: `60.0` mm/s

Infill Overlap: `15` %

Infill Flowrate: `100.0` %

Infill Pattern Type: `Grid`

Infill Extrusion Width Percentage: `100` %

Infill Offset X: `0.00` mm

Infill Offset Y: `0.00` mm

Combine Infill Layers: `1`

Top and Down Solid Part

Bottom Solid Fill Layers: `5`

Top Solid Fill Layers: `5`

Bottom Solid Fill Speed: `50.0` mm/s

Top Solid Fill Speed: `30.0` mm/s

Bottom Solid Fill Flowrate: `100.0` %

Top Solid Fill Flowrate: `100.0` %

Bottom Solid Fill Pattern Type: `Lines`

Top Solid Fill Pattern Type: `Lines`

☑ Use Gap Filling for Faild Solid Fill Parts

Infill Angle

Angle: `0` Deg

```
45
135
```

Add Infill Angle

Remove Infill Angle

Top and Down Solid Fill Angle

Angle: `0` Deg

```
45
135
```

Add Infill Angle

Remove Infill Angle

Advanced Settings

Layer | Extruder | Infill | **Support** | Platform Additions | Cooling | Temperature | Advanced | Ooze | Other | GCode

Generate Support: Touch Platform Only ▼

Support

Support Extruder:	Primary Extruder ▼
Support Type:	Normal ▼
Support Infill Type:	Lines ▼
Support Speed:	20.0 ▲▼ mm/s
Infill Ratio:	40 ▲▼ %
Max Overhang Angle:	70 ▲▼ Deg
Horizontal Offset:	0.70 ▲▼ mm
Vertical Offset Top Layers:	1 ▲▼
Vertical Offset Down Layers:	1 ▲▼
Support Flowrate:	100.0 ▲▼ %
Horizontal Expansion:	0.00 ▲▼ mm
Solid Base Layers:	0 ▲▼

Dense Support

Dense Support Layers:	0 ▲▼
Dense Support Infill Ratio:	80 ▲▼ %
Dense Support Infill Type:	Line ▼
Dense Support Extruder:	Primary Extruder ▼

Interlaced Support Infill Angle

Angle: 0 ▲▼ Deg

[Add Infill Angle]

[Remove Infill Angle]

0

Other

Pillar Size: 4.00 ▲▼ mm

☐ Add Sparse Connection

Advanced Settings

Layer | Extruder | Infill | Support | **Platform Additions** | Cooling | Temperature | Advanced | Ooze | Other | GCode

Platform Addition: None ▼

Raft

Raft Extruder:	Primary Extruder ▼
Raft Offset:	5.00 ▲▼ mm
Raft Gap from Model:	0.20 ▲▼ mm
Raft Lines Type:	Lines ▼

☐ Keep Holes in Raft Structure

First Layer | Middle Layer | Surface Layer

First Layers:	2 ▲▼
First Layer Speed:	8.0 ▲▼ mm/s
Extrusion Width Percentage:	200 ▲▼ %
Layer Height:	0.5000 ▲▼ mm
First Layer Infill Ratio:	33 ▲▼ %
First Layer Infill Angle:	0 ▲▼ Deg

Skirt and Brim

Skirt/Brim Extruder:	Primary Extruder ▼
Skirt Loop Lines:	4 ▲▼
Skirt Offset Distance:	5.0 ▲▼ mm
Brim Loop Lines:	5 ▲▼

☐ Print Outer Shell Before Inner Shell in the First Layer

Advanced Settings

Tabs: Layer | Extruder | Infill | Support | Platform Additions | **Cooling** | Temperature | Advanced | Ooze | Other | GCode

Cooling

Setting	Value	Unit
Minimal Layer Print Time:	15.0	sec
☑ Slow Down Printing Speed		
Minimal Printing Speed:	10.0	mm/s
☑ Increase Fan Speed		
Maximum Fan Speed:	100	%

☐ Blip fan speed to 100% when starting from low speed

Setting	Value	Unit
Low Fan Speed Threshold:	30	%
Blip Fan Speed Pause Duration:	500	msec

Fan Control

Layer: 1
Fan Speed: 0 %

[Add Fan Point]
[Remove Fan Point]

Layer	Fan Speed
1	0
2	25
3	75
4	100

Advanced Settings

Tabs: Layer | Extruder | Infill | Support | Platform Additions | Cooling | **Temperature** | Advanced | Ooze | Other | GCode

Temperature

Setting	Value	Unit
Heated Bed Temperature:	50	°C
Primary Extruder:	205	°C

☐ Use Temperature Control List

Tabs: Heated Bed | Primary Extruder

Layer: 1
Temperature: 40 °C

[Add Temperature]
[Remove Temperature]

Layer	Temperature

☐ Cool Down Inactive Extruder

Setting	Value	Unit
Park Position X:	30.00	mm
Park Position Y:	295.00	mm
Inactive Cooling Temperature (Left):	180	°C
Inactive Cooling Temperature (Right):	180	°C
☐ Heat up Inactive Extruder in Advance		
Heat up Ahead of Time:	40.0	sec
Inactive Heating Temperature (Left):	200	°C
Inactive Heating Temperature (Right):	200	°C

Advanced Settings

Tabs: Layer | Extruder | Infill | Support | Platform Additions | Cooling | Temperature | **Advanced** | Ooze | Other | GCode

☑ Enable Acceleration Control   ☑ Enable Jerk Control

Acceleration

Setting	Value	Unit
Printing Acceleration:	400.00	mm/s2
Inner Shell Acceleration:	400.00	mm/s2
Outer Shell Acceleration:	400.00	mm/s2
Infill Acceleration:	400.00	mm/s2
Bottom Solid Fill Acceleration:	400.00	mm/s2
Top Solid Fill Acceleration:	400.00	mm/s2
Travel Acceleration:	400.00	mm/s2

Jerk

Setting	Value	Unit
Printing Jerk:	12.00	mm/s
Inner Shell Jerk:	12.00	mm/s
Outer Shell Jerk:	12.00	mm/s
Infill Jerk:	12.00	mm/s
Bottom Solid Fill Jerk:	12.00	mm/s
Top Solid Fill Jerk:	12.00	mm/s
Travel Jerk:	12.00	mm/s

☐ Override Filament Settings

Primary Filament Flowrate: 100.0 %

Advanced Settings

Tabs: Layer | Extruder | Infill | Support | Platform Additions | Cooling | Temperature | Advanced | **Ooze** | Other | GCode

Other

☑ Force Retraction On Layer Change

☐ Force Retraction Before Traveling to Outer Shell

☑ Avoid Retraction Inside Models

☑ Enable Retraction at Bottom and Top of Models

Bottom Layers: 1

Top Layers: 2

Extruder Switch Ooze Control

Setting	Value	Unit
Retraction Speed of Extruder-switch:	20.0	mm/s
Retraction Amount of Extruder-switch:	3.50	mm
Restart Speed of Extruder-switch:	20.0	mm/s
Extra Restart Amount of Extruder-switch:	0.00	mm

Travel

☐ Avoid Traveling Through Holes

☐ Maximum Travel Path Length: 200 %

☐ Wipe Wall

Setting	Value	Unit
Wipe Wall Offset:	2.00	mm
Wipe Wall Angle:	30	Deg
Wipe Wall Loop Lines:	1	
Wipe Wall Type:	Contoured	
Wipe Wall Speed:	50.0	mm/s

☐ Wipe Tower

Setting	Value	Unit
Wipe Tower Width:	12.00	mm
Wipe Tower Infill Ratio:	100	%
Wipe Tower Speed:	50.0	mm/s

☐ Placed at the Fixed Position of the Platform

Setting	Value	Unit
Wipe Tower Offset X:	-2.00	mm
Wipe Tower Offset Y:	2.00	mm
Wipe Tower Fixed Position X:	25.00	mm
Wipe Tower Fixed Position Y:	280.00	mm

Advanced Settings

Tabs: Layer | Extruder | Infill | Support | Platform Additions | Cooling | Temperature | Advanced | Ooze | **Other** | GCode

Repair

- [ ] Merge Open Segments of Model Parts
- [ ] Merge Internal Overlapping Parts

Thin Wall

- [x] Check Thin Wall (Single Extrusion Width)

Minimal Extrusion Width Percentage: `25` %

Maximum Extrusion Width Percentage: `200` %

Gap Filling

- [x] Fill Gaps in Shells

Gap Filling Speed: `60.0` mm/s

- [ ] Allow Filling Gaps using Single Extrusion Filling

Minimal Single Extrusion Width Percentage: `33` %

Maximum Single Extrusion Width Percentage: `200` %

Single Extrusion Filling Speed: `25.0` mm/s

Pause at Height

Height: `5.000` mm

[Add Pause Point]

[Remove Pause Point]

The height property does not include raft.

- [x] Enable Bridging Detection

Extrusion Width Percentage: `100` %

Bridging Speed: `60.0` mm/s

- [ ] Enable Bridging Fan Speed `100` %

- [ ] Enable Fixed Bridging Angle `0` Deg

Minimal Allowed Briding Area: `20.00` mm2

Global Offset

X-Offset: `0.00` mm

Y-Offset: `0.00` mm

Z-Offset: `0.00` mm

Advanced Settings

Tabs: Layer | Extruder | Infill | Support | Platform Additions | Cooling | Temperature | Advanced | Ooze | Other | **GCode**

Sub-tabs: **Start GCode** | End GCode | Extruder Switch GCode | Layer Change GCode | Retraction GCode

Start GCode when the number of extruders is `1`

```
G21
G90
M82
M107
G28 X0 Y0
G28 Z0
G1 Z15.0 F{travel_xy_speed}
G92 E0
G1 F140 E29
G1 X20 Y0 F140 E30
G92 E0
G1 F{travel_xy_speed}
M117 Printing...
```

- [x] Insert Heatup GCode Automatically in Start GCode

Heatup Sequence: `Heatup heated bed and nozzle(s) simultaneously`

Built-In Placeholders:

(Double click item to add to gcode)

- ∨ Slice Settings
  - {travel_xy_speed}
  - {layer_default_speed}
  - {retraction_speed1}
  - {temperature_extruder1}
  - {temperature_extruder2}
  - {temperature_heatbed}
  - {extruder_switch_retraction_amount}

[Add More] [Remove Selected]

Post-process Commands

[Add] [Remove]

(Double click item to edit settings)