

Clint's Slicer Settings

This document describes the release package for Clint's Slicer Settings – a package of settings for 3D printer slicer software that converts 3D models in STL format to lower-level commands for particular FDM hardware (typically 3D printers). The package is currently specific to Simplify3D (S3D) version 4.0.1 and the Prusa i3 Mk3 printer (Pi3Mk3) with a 0.40 mm nozzle. It contains two S3D “Profile” (.fff) files with slicer settings for:

- General (i.e. not highly-modified) PLA, and
- ColorFabb's nGen (or nGen Lux), a filament made from Eastman's Amphora AM3000 co-polyester.

While these slicer settings were specifically designed for use with the Breath Flute project (www.BreathFlute.com), the S3D profiles are appropriate for use as “core” profiles for general use.

These profiles were developed over many months from the 12 core profiles I assembled into the Simplify3D Profiles Comparison document and profiles, available in a ZIP archive at http://www.BreathFlute.com/zip/S3D_Pi3Mk3_CoreProfiles_CG.zip.

You may also be interested in the document at http://www.BreathFlute.com/pdf/S3D_SettingsCG.pdf, which provides a collection of documentation on the myriad of S3D parameters and settings.

The latest version of document and its associated profiles are available in the release package / ZIP archive located at http://www.BreathFlute.com/zip/ClintGoss_SlicerSettings.zip.

I hope this is helpful!

— Clint Goss, Ph.D. [clint@goss.com], 8/26/2018

Background

The AutoConfigure settings for Quality in these profiles are unusual. They are designed for printing “Main” and “Tool” components of the Breath Flute project, and have options for various Additions (Skirt, No-Skirt, and Brims of 10 and 16 outlines).

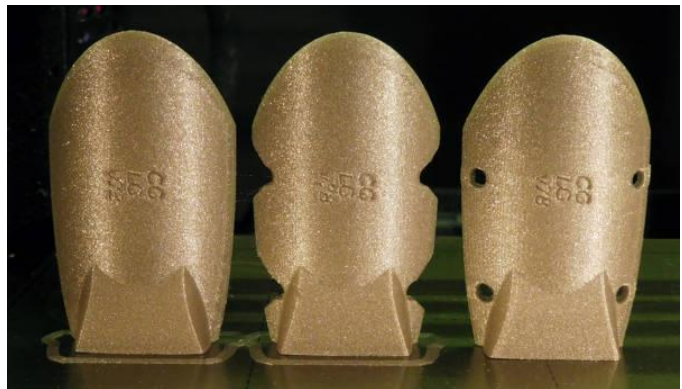
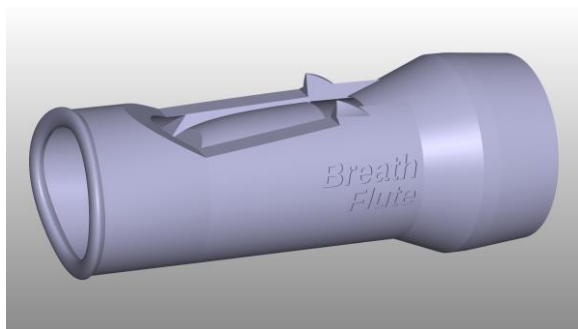
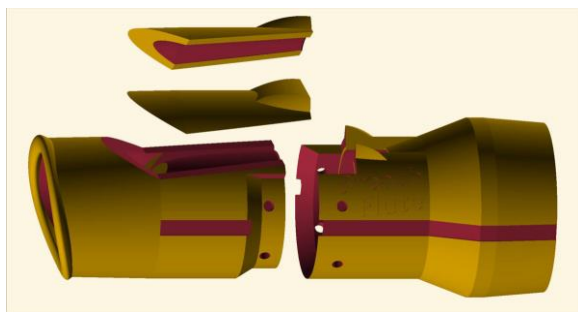
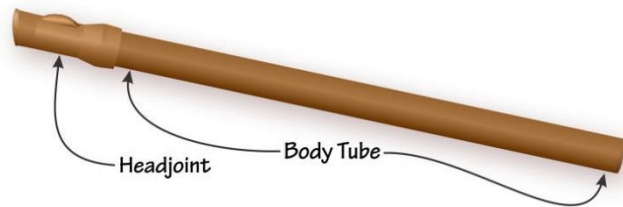
- The “Main” components of the Breath Flute are the primary components that go into the musical instrument. These are available for PLA and nGen at a Layer Height of 100 µm (0.1 mm).
- The “Tool” components are specific tools for post-processing (i.e. sanding) the main components. These are available for PLA at a Layer Height of 200 µm (0.2 mm).

For other settings, you can edit the .fff files in a text editor (Notepad, vi, etc) or load them into Simplify3D and examine a specific Auto-Configure variant.

Note that the names of the .fff files as well as the names of the profiles (contained in XML fields in the .fff files) have been altered to provide better identification and organization.

Impetus

This work was done as part of my main 3D printing effort – to develop a new style of flute called the **Breath Flute**. The flute consists of a 3D-printed headjoint and four feet of 1¼" tube. The OpenSCAD model currently runs about 11,000 lines of code (with comments). This page has some images from the current state of development. More information is available at: www.BreathFlute.com.



Copyright © 2018 Clinton F. Goss. This document is part of Clint's Slicer Settings, available as a stand-alone release package at www.BreathFlute.com. It is also released as part of the Breath Flute Project release package.

You may copy, use, modify, and distribute this document under the terms of the *GNU Free Documentation License* version 1.3 ("the *GFDL*") or any later version as published by the Free Software Foundation; with no Invariant Sections, no Front-Cover Texts, and no Back-Cover Texts.

This document is distributed in the hope that it will be useful, but **WITHOUT ANY WARRANTY**; without even the implied warranty of **MERCHANTABILITY** or **FITNESS FOR A PARTICULAR PURPOSE**. See the *GFDL* for more details.

The *GFDL* in plain text is distributed with releases of Clint's Slicer Settings. It is also available at <http://www.GNU.org/>.

Clint Goss Pi3Mk3 Breath Flute-Specific PLA Profile

ClintGoss_S3D_401_Pi3Mk3_PLA_20180826.fff

```
File:          ClintGoss_S3D_401_Pi3Mk3_PLA_20180826.fff
Author:        Clint Goss
Created:       8/19/2018
Released on:   8/20/2018
Available at:  http://www.BreathFlute.com/zip/ClintGoss_SlicerSettings.zip
Src Dir:       /BreathFlute/S3D_Profiles/ClintGoss_SlicerSettings
Download Pkg:  ClintGoss_SlicerSettings.zip
Src File:      ClintGoss_S3D_401_Pi3Mk3_PLA_20180826.fff
Orig XML Prof: <profile name="CG_Pi3Mk3_BF_PLA" version="2018-08-19 10:00:00" app="S3D-Software 4.0.1">
Orig XML Base: <baseProfile>CG_Pi3Mk3_BF</baseProfile>
Core XML Prof: <profile name="ClintGoss_S3D_401_Pi3Mk3_PLA_20180826"
               version="2018-08-26 10:00:00" app="S3D-Software 4.0.1">
Core XML Base: <baseProfile>CG_Pi3Mk3_BF_PLA</baseProfile>
```

On 5/2/2018, I began work on an S3D profile that is specifically for use with prints of the Breath Flute. It is based on the core profile S3DProfile_Pi3Mk3_S3D_Pristine_20180412_Core.fff / PLA / High. Here are all the parameters of that profile, with differences from the core profile – S3DProfile_Pi3Mk3_S3D_Pristine_20180412_Core.fff – are shown in red.

Profile: ClintGoss_S3D_401_Pi3Mk3_PLA_20180826

AutoConfigureMaterial: PLA

AutoConfigureQuality: Main-Skirt, Main-NoSkirt, Main-Brim10, Main-Brim16,
Tool-Skirt, Tool-NoSkirt, Tool-Brim10, Tool-Brim16

Extruder: E-List [PrimExtr], Index Tool 0: Noz 0.40, ExtMult 1.00, ExtWid Man 0.40,
Ooze Control: YES Retr, RetrDist 1.00, ExRestart 0.00, RVertLift 0.00,
RSpeed 2400 mm/min (→ 40 mm/sec), Yes Coast, CoastDist 0.20, Yes Wipe, WipeDist 2.00

Layer: L-Extr: PrimExtr,

*// Layer Height 0.1mm = 100um – good for visual quality. However, Infill=>Combine set to 2
// to print infill at 0.2mm, which has 24% better Yield Stress, [3DMATTER 2015]*

LHt 0.10 (Quality=Main-*), 0.20 (Quality=Tool-*),

TSolid 4, BSolid 4,

Shells 3, *// Highest Yield Stress, based on the [3DMATTER 2015] study*

Dir: InOut, No PISeq, No Vase,

FHt: 150%, *// S3D Default ... works well for me. Jo Prusa & Jeff Golden use 200%*

FWid 105%, *// A little extra extrusion to aid in sticking to the bed. Chris Warkocki uses 102%*

FSpeed 35% (1680 mm/min → 28 mm/sec), *// Warkocki uses 25 mm/sec, other experienced use 30 mm/sec
// Start on the back. For the Bird component, the seam will be in critical areas (roof of the flue,
// top curved bevel edge), but those areas require sanding in any case.*

// Issue: As we move the print around on the bed to handle PEI bubbles ... this needs to be adjusted!

StartPts: Choose, X:125.0 / Y:210.0 (Quality=Main-*), RandomStart (Quality=Tool-*),

Additions: Skirt: No (Quality=*-NoSkirt) otherwise Yes,
Sk-Extr PrimExtr, SkLayers 1,
SkOffset 3.00 (Quality=*-Skirt), 0 (Quality=*-Brim*),
SkOutlines 2 (Quality=*-Skirt), 10 (Quality=*-Brim10), 16 (Quality=*-Brim16),
No Raft, R-Extr PrimExtr, Top 3, Base 2, ROffset 3.00, SepDist 0.14, TopInfill 100%,
SpAbRaft 30% (@SpDefault of 4800 mm/min → 1440 mm/min → 24 mm/sec),
No Pillar, P-Extr AllExtr, PPWidth 12.00, PPLoc North-West,
PPSpMult 100% (@SpDefault of 4800 mm/min → 4800 mm/min → 80 mm/sec),
No Ooze O-Extr AllExtr, OOffset 2.00, OPerims 1, OShape Waterfall, OAngle 30°,
OSpMult 100% (@SpDefault of 4800 mm/min → 4800 mm/min → 80 mm/sec)

Infill: I-Extr PrimExtr,
IntPat Rect, // Rectilinear for greatest Max Stress, based on the [3DMATTER 2015] study
// Concentric, so that visible surfaces show arcs rather than lines
ExtPat Concentric (Quality=Main-*), Rectilinear (Quality=Tool-*),
// 90% Infill for greatest Yield Stress, based on the [3DMATTER 2015] study
Infill 90% (Quality=Main-*), 30% (Quality=Tool-*),
OutOvr 25%, // S3D is 20%, KeyboardWarrior 30%, Jo Prusa 25%
InWid 100%,
MinLen 2.00, // S3D is 5, Jo Prusa is 2, Chris Warkocki is 1.00
// For MAIN parts at 0.1mm layer height, force infill to print at 0.2mm, which increases
// Yield Stress by 24% [3DMATTER 2015]
Combine 2 (Quality=Main-*), 1 (Quality=Tool-*),
No IncSolid, DiaphEvery 20,
// For MAIN parts, Best Max Stress along longitudinal axis, [3DMATTER 2015]
IntAng: 0 / +50 / -25 / +25 / -50 / 90 (Quality=Main-*), 45 / -45 (Quality=Tool-*),
No EvAngle,
ExtAng: 45 / -45 // This should not have any effect on slicing if ExtPat is Concentric

Support: No GenSupp, Sp-Extr PrimExtr, SuppInfill 40%, ExInfilDist 0.00, BaseLayers 0, CombEvery 1,
DenseSupp D-Extr PrimExtr, DenseLayers 0, DenseInfill 70%,
APType Normal, APRes 4.00, APAngle 45°, HSep 0.30, UpLay 1, LowLay 1, SupAng 0

Temp: T-List [PrimExtr] T0, TType Extruder, No Layer, No Loop, Yes WaitStab,
SetP 1:210°C / 2:205°C, // Somewhat arbitrary setting for PLA
T-List [HBed] T0, TType HBPlat, No Layer, No Loop, Yes WaitStab,
SetP 1:65°C / 2:55°C // Somewhat arbitrary setting for PLA

Cooling: FanSpeed 1:0 / 2:100,
No Blip, No Incr, ITime 45sec, MxFSp 100%, No BrFan, BrSpOvr 100%

G-Code: Yes 5D, No RelDist, Yes AllowZ, No Indep, No M101, Yes Sticky, No applyGOffsets,
GOffsets X:0.00 / Y:0.00 / Z:0.00, Yes UpMDef, MType Cart, Build X:250 / Y:210 / Z:210,
Orig X:0 / Y:0 / Z:0, Home X:Min / Y:Min / Z:Min, Flip No X, Yes Y, No Z,
THeadOffsets [Tool 0] X:0 / Y:0,
Yes UpdFirm, FType RepRap, GPX: Replicator 2, Baud 115200

Scripts:

Starting Script:

```
M115 U3.0.7 ; use the latest firmware version
G28 W ; home all axes without mesh bed leveling
G80 ; run mesh bed leveling routine
G1 Y-3.0 F1000.0 ; prepare to prime
G92 E0 ; reset extrusion distance
G1 X60.0 E9.0 F1000.0 ; priming
G1 X100.0 E12.5 F1000.0 ; priming
```

Ending Script:

```
G4 ; wait
M106 S0 ; turn off cooling fan
M104 S0 ; turn off extruder
M140 S0 ; turn off bed
G1 X0 Y200 ; part removal
M84 ; disable motors
```

Other scripts are blank.

ExpFmt: Standard G-Code, No AddCeleb, AddTermCmd: (none)

```
// Max Print Speed = Max Volume / (Extrusion Width × Layer Height)
//                      = 10 mm³/sec / (0.4 mm × 0.1 mm)
//                      = 250 mm/sec
// Min Print Speed = 6 mm/sec (based on "Bulge Jam" scenario)
```

Speeds: SpDefault 4800 mm/min (→ 80 mm/sec), // *Core profiles range 1,800–12,000 mm/min*
SpPerim 50% (@SpDefault of 4800 mm/min → 2400 mm/min → 40 mm/sec),
SpSolidIn 80% (@SpDefault of 4800 mm/min → 3840 mm/min → 64 mm/sec),
SpSupp 80% (@SpDefault of 4800 mm/min → 3840 mm/min → 64 mm/sec),
SpXY 4800 mm/min (→ 80 mm/sec), // *S3D was 12000. Don't really need to go blazing around ...*
SpZ 600 mm/min (→ 10 mm/sec), // *S3D was 1000. Be kind to your Z axis ...*
Yes AdjBelow, AdjBelowSec 15,
AdjDown 30% (@SpDefault of 4800 mm/min → 1440 mm/min → 24 mm/sec,
@SpPerim of 2400 mm/min → 720 mm/min → 12 mm/sec)

Other: Area 50.0,

ExInflat 1.00, *// Extra support for bridging (Warkocki & Jo Prusa profiles)*

BrExMult 100%, BrSpMult 100%, No FixedAngle 0°,

Yes BrPerim, *// This seems like a smart idea ... not sure why S3D has it off*

HComp 0.00, F-List [Tool 0] FilDiam 1.75, FilPrice 46.00, FilDen 1.25,

ChgRetDist 12.00, ChResDist -0.50, ChRetSp 600

Advanced: No Start 0.00, No Stop 0.00,

// Tried this, but it failed badly at the top (proximal) edge of the printed bird ... extruded a bunch of "grit".

// HOWEVER ... need to address how this could be used for the splitting edge – where we might

// want a single-wall extrusion.

// ExThinType AllowSingWall, // Allow a wall with only a single extrusion

ExThinType PerimOnly, *// Force full perimeters to be printed (S3D Default)*

InThinType AllowSingExtr, *// Allow internal fills using a variable-width single extrusion*

Overlap 40%, *// Permit more combining of infills (default is 25%)*

MinExLen 1.00, MinPWid 50%, MaxPWid 200%,

EndExtDist 0.40, *// S3D was 0.20. Naver blog suggestions "usually = nozzle size"*

Yes Open, Yes ForceRet, No MinTRetr 3.00, Yes RetWipe, Yes WipeOuter,

Yes AvoidCross, *// Seems like a good idea*

MaxDetour 5.0, *// S3D was 3.0. Jo Prusa uses 5.0*

NonManSeg Heal, No Merge

Guide to Printing with Amphora AM3300

This advice on printing in filaments made from AM3300 is from the document:

Eastman, *Guide to Printing with Amphora AM3300*, by Eastman, SP-MBS-1803, last modified 5/15/2016, retrieved 5/18/2017 from https://www.eastman.com/Literature_Center/S/SPMBS1803.pdf.

Similar advice can be found in the *Amphora 3D Polymer Consumer Selector Guide* by Eastman, SP-MBS-4044, 8 pages, last modified 2/9/2017, initially retrieved 5/18/2017 from https://www.eastman.com/Literature_Center/S/SPMBS4044.pdf (but still current as of 5/9/2018).

Suggested settings from this document are marked with *// AmphoraCSG*.

The following recommendations have been established for optimal Amphora AM3300 printing conditions.

Temperature: 220°–250°C. Start at 220°C and increase 5° until desired flow and proper layer adhesion are established. Different printers may require slightly different temperature settings.

Heated bed: 65°–85°C. Bed adhesion will be achieved when using a clear glass plate; however, there is a risk of pulling glass pieces from the build surface when using this method. Hair spray can be used to reduce the risk of the print adhering to the build bed.

Percent (%) flow: 90%–104%. Set first layer height at 150% with a width of 120% and 40% speed. Start the print with a skirt to ensure optimal filament flow through the nozzle.

Retraction distance: 3mm. Retraction distance varies from printer to printer. Use suggested retraction distance with a 0.3 mm coast and 0.3 mm vertical lift.

Retraction speed: 45 mm/sec. Experiment with your printer as necessary to determine optimal retraction settings.

Printing speed: 35–50 mm/sec. Do not print too fast. Allow the material time to effectively stick to the build plate and first layers.

Layer height: 0.15–0.20mm on a 0.4mm nozzle. Monitor the first layer to make sure no material sticks to the nozzle and accumulates. If the first layer has no buildup or skips a layer the rest of the print should be successful.

Cooling fan: Engage in cooling at 60% after layer 4. This percent varies from printer to printer. Too much cooling combined with low print temperature and high print speed could result in layer delamination in prints.

Best bridging: 60 mm/sec with 100% cooling. Aim to run the fan at 100% rate during bridging operations when attempting to print overhangs or gaps without physical support. If bridging is done too quickly, melt break will occur. However, if done too slowly, layers will start sagging.

Finish: Cool bed to less than 40°C. Minimize print distortion and allow easier removal by cooling before attempting to remove the print from the bed.

Clint Goss S3D Pi3Mk3 Breath Flute-Specific nGen Profile

ClintGoss_S3D_401_Pi3Mk3_nGen_20180826.fff

```
File:          ClintGoss_S3D_401_Pi3Mk3_nGen_20180826.fff
Author:        Clint Goss
Created:       8/19/2018
Released on:   8/20/2018
Available at:  http://www.BreathFlute.com/zip/ClintGoss_SlicerSettings.zip
Src Dir:       /BreathFlute/S3D_Profiles/ClintGoss_SlicerSettings
Download Pkg:  ClintGoss_SlicerSettings.zip
Src File:      ClintGoss_S3D_401_Pi3Mk3_nGen_20180826.fff
Orig XML Prof: <profile name="CG_Pi3Mk3_BF_nGen" version="2018-08-22 06:03:37" app="S3D-Software 4.0.1">
Orig XML Base: <baseProfile>CG_Pi3Mk3_BF</baseProfile>
Core XML Prof: <profile name="ClintGoss_S3D_401_Pi3Mk3_nGen_20180826"
               version="2018-08-26 10:00:00" app="S3D-Software 4.0.1">
Core XML Base: <baseProfile>CG_Pi3Mk3_BF_nGen</baseProfile>
```

Items shown in **purple-bold** are changes from the PLA profile above.

Profile: ClintGoss_S3D_401_Pi3Mk3_nGen_20180826

AutoConfigureMaterial: ColorFabb nGen / AM3300

AutoConfigureQuality: Main-Skirt, Main-NoSkirt, Main-Brim10, Main-Brim16

Extruder: E-List [PrimExtr], Index Tool 0: Noz 0.40,

// AmphoraCSG: 90%–104% Set first layer at 150% [?is that Fht?] with a width of 120% and 40% speed.

ExtMult 1.00,

// Increased from 1.00×Noz to 1.15×Noz based on the use of a 1.2 factor as a default for S3D and Prusa

// forum user PJR that “generally extrusion width should be from 1 to 1.25 times the nozzle diameter”

// (May 2, 2018). See S3D_SettingsCG.pdf for more info.

ExtWid Man 0.46, *// Up from 0.40 in PLA*

Ooze Control: YES Retr,

// AmphoraCSG: 3mm Use suggested retraction distance with a 0.3 mm coast and 0.3 mm vertical lift.

RetrDist 3.00, *// Up from 1.00 in PLA*

ExRestart 0.00,

// AmphoraCSG – from the Retraction Distance reco: ... with a 0.3 mm coast and 0.3 mm vertical lift.

RVertLift 0.30, *// Up from 0.00 in PLA*

// AmphoraCSG: 45 mm/sec (→ 2700 mm/min)

RSpeed 2700 mm/min (→ 40 mm/sec), *// Up from 2400 in PLA*

// AmphoraCSG – from the Retraction Distance reco: ... with a 0.3 mm coast and 0.3 mm vertical lift.

Yes Coast, **CoastDist 0.30,** *// Up from 0.20 in PLA*

Yes Wipe, WipeDist 2.00

Layer: L-Extr: PrimExtr,

// Layer Height 0.1mm = 100um – good for visual quality. However, Infill=>Combine set to 2

// to print infill at 0.2mm, which has 24% better Yield Stress, [3DMATTER 2015]

// AmphoraCSG: 0.15–0.20 mm. Monitor the first layer to make sure no material sticks to the nozzle

// and accumulates. If the first layer has no buildup or skips in the layer, the rest of the print should

// be successful. Height ranges are for a 4 mm [sic] nozzle.

LHt 0.10, // BELOW the minimum AmphoraCSG recommendation.

TSolid 4, BSolid 4, Shells 3, // Shells=3 for highest Yield Stress, based on the [3DMATTER 2015] study

Dir: InOut, No PISeq, No Vase,

// AmphoraCSG – from the Percent Flow reco: Start first layer at 150% [?is that FHt?]

FHt: 150%, // S3D Default ... works well for me. Jo Prusa & Jeff Golden use 200%

// AmphoraCSG – from the Percent Flow reco: ... with a width of 120%

FWid 120%, // Up from 105% in PLA. Chris Warkocki uses 102%

// AmphoraCSG – from the Percent Flow reco: ... and 40% speed. Up from 35% in PLA

FSpeed 40% (@SpDefault of 3000 mm/min → **1200 mm/min** → **20 mm/sec**),

// Start on the back. For the Bird component, the seam will be in critical areas (roof of the flue,

// top curved bevel edge), but those areas require sanding in any case.

// Issue: As we move the print around on the bed to handle PEI bubbles ... this needs to be adjusted!

StartPts: Choose, X:125.0 / Y:210.0,

Additions: Skirt: No (Quality=*-NoSkirt) otherwise Yes,

Sk-Extr PrimExtr, SkLayers 1,

SkOffset 3.00 (Quality=*-Skirt), 0 (Quality=*-Brim*),

SkOutlines 4 (Quality=*-Skirt), 10 (Quality=*-Brim10), 16 (Quality=*-Brim16),

No Raft, R-Extr PrimExtr, Top 3, Base 2, ROffset 3.00, SepDist 0.14, TopInfill 100%,

SpAbRaft 30% (@SpDefault of 3000 mm/min → **900 mm/min** → **15 mm/sec**),

No Pillar, P-Extr AllExtr, PPWidth 12.00, PPLoc North-West,

PPSpMult 100% (@SpDefault of 3000 mm/min → **3000 mm/min** → **50 mm/sec**),

No Ooze O-Extr AllExtr, OOffset 2.00, OPerims 1, OShape Waterfall, OAngle 30°,

OSpMult 100% (@SpDefault of 3000 mm/min → **3000 mm/min** → **50 mm/sec**)

Infill: I-Extr PrimExtr,

IntPat Rect, // Rectilinear for greatest Max Stress, based on the [3DMATTER 2015] study

ExtPat Concentric, // Concentric, so that visible surfaces show arcs rather than lines

Infill 90%, // 90% Infill for greatest Yield Stress, based on the [3DMATTER 2015] study

OutOvr 25%, // S3D is 20%, KeyboardWarrior 30%, Jo Prusa 25%

InWid 100%,

MinLen 2.00, // S3D is 5, Jo Prusa is 2, Chris Warkocki is 1.00

Combine 2, // Force infill to print at 0.2 mm – increases Yield Stress by 24% [3DMATTER 2015]

No IncSolid, DiaphEvery 20,

IntAng: 0 / +50 / -25 / +25 / -50 / 90, // Best Max Stress along longitudinal axis, [3DMATTER 2015]

No EvAngle,

ExtAng: 45 / -45 *// This should not have any effect on slicing if ExtPat is Concentric*

Support: No GenSupp, Sp-Extr PrimExtr, SuppInfill 40%, ExInflDist 0.00, BaseLayers 0, CombEvery 1, DenseSupp D-Extr PrimExtr, DenseLayers 0, DenseInfill 70%,
APType Normal, APRes 4.00, APAngle 45°, HSep 0.30, UpLay 1, LowLay 1, SupAng 0

Temp: T-List [PrimExtr] T0, TType Extruder, No Layer, No Loop, Yes WaitStab,
// AmphoraCSG: 220°–250°C Start at low temperature and increase 5°C
// until desired flow and proper adhesion is established.
SetP 1:210°C / 2:205°C,

T-List [HBed] T0, TType HBPlat, No Layer, No Loop, Yes WaitStab,
// AmphoraCSG: 65°–85°C For best adhesion, use clear glass plate. Applying hair spray
// reduces the risk of pulling glass pieces from the surface.
SetP 1:65°C / 2:55°C

Cooling:

// AmphoraCSG: Engage cooling at 60% after layer 4. Recommended for regular print jobs requiring
// an average strength and speed. For best possible strength, it is advised to print with as little cooling
// as possible. This percent varies from printer to printer. Be careful using too much cooling: this
// combined with low print temperature and high speed could result in layer delaminating prints.

FanSpeed 1:0 / **4:60**, *// From 1:0 / 2:100 in PLA*

No Blip, No Incr, ITime 45sec, MxFSp 100%,

// AmphoraCSG: Plan to run the fan at 100% during bridging operations when attempting to
// print overhangs or gaps without physical support. If bridging is done too quickly, break melt
// will occur. However, if done too slowly, layers will start sagging.

Yes BrFan, BrSpOvr 100%

G-Code: Yes 5D, No RelDist, Yes AllowZ, No Indep, No M101, Yes Sticky, No applyGOffsets,
GOffsets X:0.00 / Y:0.00 / Z:0.00, Yes UpMDef, MType Cart, Build X:250 / Y:210 / Z:210,
Orig X:0 / Y:0 / Z:0, Home X:Min / Y:Min / Z:Min, Flip No X, Yes Y, No Z,
THeadOffsets [Tool 0] X:0 / Y:0,
Yes UpdFirm, FType RepRap, GPX: Replicator 2, Baud 115200

Scripts: All scripts are identical to the PLA profile above.

ExpFmt: Standard G-Code, No AddCeleb, AddTermCmd: (none)

```
// Max Print Speed = Max Volume / (Extrusion Width × Layer Height)
//                      = 10 mm³/sec / (0.4 mm × 0.1 mm)
//                      = 250 mm/sec
// Min Print Speed = 6 mm/sec (based on "Bulge Jam" scenario)
```

Speeds:

*// AmphoraCSG: 40–60 mm/sec (→2400–3600 mm/min) Printing speed is typically similar to
 // PLA setting on most machines.*

// Core profiles range 1,800–12,000 mm/min

SpDefault 3000 mm/min (→ **50 mm/sec**), *// Down from 4800 mm/min PLA*

SpPerim 50% (@SpDefault of 3000 mm/min → **1500 mm/min** → **25 mm/sec**),

SpSolidIn 80% (@SpDefault of 3000 mm/min → **2400 mm/min** → **40 mm/sec**),

SpSupp 80% (@SpDefault of 3000 mm/min → **2400 mm/min** = **40 mm/sec**),

// SpXY changed 4800 → 12000 to combat stringing.

// This is similar to the advice for ColorFabb Corkfill, another stringing-prone filament, from

// <https://learn.colorfabb.com/how-to-print-with-corkfill/>, retrieved 5/6/2018:

// With a 0.2mm Layer Height, we found a good balance between time and surface quality.

// Little tip: Check your travel speed! We also recommend to use a high travel speed of at least

// 200mm/s to prevent stringing.

// Also, advice from Tomas Sanladerer: (<https://toms3d.org/2018/02/05/things-know-petg/> and

// https://www.youtube.com/watch?v=8_adY2K-YIc

// "Travel moves should always be as fast as your printer can handle, because it's going to give the

// hotend as little of a chance of oozing and going out of the intended plastic flow situation as possible

// if that makes sense. Your printer's firmware should know how fast it can reliably go, so you can

// actually set the travel speed on your slicer to some obscene value and let your printer handle the rest."

SpXY 12000 mm/min (→ **200 mm/sec**),

SpZ 600 mm/min (→ 10 mm/sec), Yes AdjBelow, AdjBelowSec 15,

AdjDown 30% (@SpDefault of 3000 mm/min → **900 mm/min** → **15 mm/sec**,

@SpPerim of 1500 mm/min → **450 mm/min** → **7.5 mm/sec** ...

close to the 6 mm/sec Min Print Speed!)

Other: Area 50.0,

ExInflat **1.00**, *// Extra support for bridging (Warkocki & Jo Prusa profiles)*

BrExMult 100%,

// AmphoraCSG: 60 mm/sec. If bridging is done too quickly, break melt will occur. However, if done

// too slowly, layers will start sagging. Up from 100% in PLA

BrSpMult 120% (@SpDefault of 3000 mm/min → **3600 mm/min** → **60 mm/sec**),

No FixedAngle 0°,

Yes BrPerim, *// This seems like a smart idea ... not sure why S3D has it off*

HComp 0.00, F-List [Tool 0] FilDiam 1.75, FilPrice 46.00, FilDen 1.25,

ChgRetDist 12.00, ChResDist -0.50, ChRetSp 600

Advanced: No Start 0.00, No Stop 0.00,
ExThinType PerimOnly, *// Force full perimeters to be printed (S3D Default)*
InThinType AllowSingExtr, *// Allow internal fills using a variable-width single extrusion*
Overlap 40%, *// Permit more combining of infills (default is 25%)*
MinExLen 1.00, MinPWid 50%, MaxPWid 200%,
EndExtDist 0.40, *// S3D was 0.20. Naver blog suggestions "usually = nozzle size"*
Yes Open, Yes ForceRet, No MinTRetr 3.00, Yes RetWipe, Yes WipeOuter,
Yes AvoidCross, *// Seems like a good idea*
MaxDetour 5.0, *// S3D was 3.0. Jo Prusa uses 5.0*
NonManSeg Heal, No Merge