3D Printer Description File Format Specification

Version 1.0 draft 3
Adobe Systems Incorporated

Introduction

3D printer profile files (3PP files) are XML files that provide a uniform approach to describing the diverse features of devices that do 3D printing. Often these printers are ultimately driven by G-code or M-code interpreters, although that is not a necessary prerequisite. Such features include different area/volume sizes, different methods of material/filament handling, memory size, and other features to be outlined. All devices do not have the same set of features, and even devices with the same features do not necessarily invoke those features in the same way. 3PP files provide applications with the necessary information about a device's features that are required to make a successful print.

Structure

There are 3 main sections of the 3PP file. The first is the "Printer" section which describes the printer itself. The printer section contains two subsections for "Quality" and "Supports".

The second section is the "Materials" section. This section contains information about each material the printer supports. While some printers may support only one kind of material; others can support hundreds. Each material may behave differently when printed such as melting points, minimum thickness, and more. This section also allows you to define the preview colors for these materials, to better differentiate them in the Photoshop UI.

The third section is "PrintPlateLevelling" which describes the bed/plate levelling controls, head positions to test, and the order in which the user should test them. This section has two subsections for "Rivets" and "Checkpoints".

```
<?xml version="1.0" encoding="UTF-8" standalone="no"?>
<Printer>
...
        <Quality>
        ...
        </Quality>
        <Supports>
```

```
        ...
        </Supports>
</Printer>
<Materials>
...
</Materials>
<PrintPlateLevelling>
...
        <Rivets>
        ...
        </Rivets>
        <Checkpoints>
        ...
        </Checkpoints>
</PrintPlateLevelling>
```

Printer Required Keywords

Version (number) - the version of the 3PP file format. Current version is 1.
`<Version value ='1' />`

Print Location (string) - defines if the printer is locally connected, remote, or accessed through an online service. Currently allowed values are "Local", "File", and "Shapeways.com". (default is File)
`<PrintLocation value='Local' />`

Print Volume (mm) - The dimensions (in millimeters) of the maximum printable volume of the printer. The X and Y dimension are in the plane of the print bed and the Z direction is up. (default is 133,133,133)
`<PrintVolume X='133' Y='133' Z='133' />`

Max Coordinates (mm) - The maximum positions of the print head (which is frequently different from the print volume). The X and Y dimension are in the plane of the print bed and the Z direction is up. (default is 230,230,230)
`<MaxCoordinates X='160' Y='160' Z='140' />`

Print Resolution (dots per mm) - The physical resolution of the printer. This is the smallest feature that can be printed by the printer. Often printers will have a different resolution in Z than they do in X and Y. The effective print resolution used during mesh repair is affected by selected quality (Detail Level). The resolution multipliers for high, medium and low qualities are 1.33, 0.8, and 0.33. (default is 10,10,10)
`<PrinterResolution X='3' Y='3' Z='10' />`

Printer Heads (number) - This is the number of heads in the printer. Many printers have only one head, but some have 2 and others may have 10 or more. (default is 1)
<PrinterHeads value='1' />

Preferred File Format (string) - All printers have a file format that either they (or for closed systems, the software that comes with them) supports. This property specifies the extension of that file format. The host application can look at this extension and decide if it had support for this printer based on this file format specifier. For example, valid options here include, but are not limited to "gcode", "s3g", "stl", "wrl". (default is stl)
 <PreferredFileFormat value='stl' />

Printer Optional Keywords

PrinterSupportsUSB (bool) - Can the printer be connected locally via USB or not? (default is false)
<PrinterSupportsUSB value ='1' />

Printer Firmware Signature (string) - This is a unique string to detect locally attached printers (requires USB support). This is only needed for USB printers, not needed for remote devices or services. The value should be a substring of a string returned by GCode M115 command
(http://reprap.org/wiki/G-code#M115:_Get_Firmware_Version_and_Capabilities)
The string comparison is not case sensitive.

Following is the example of the Solidoodle 2 signature, as the Solidoodle M115 response is similar to "ok `PROTOCOL_VERSION:0.1 FIRMWARE_NAME:FiveD FIRMWARE_URL:www.solidoodle.com MACHINE_TYPE:Mendel EXTRUDER_COUNT:1`"
<PrinterFirmwareSignature value="solidoodle" />

Printer USB Identification
    If a printer supports USB, for some cases a model can be identified via one or more USB VID/PID (Vendor ID/Product ID) pairs. To specify this, use the **<USBIDs>** node. Within the node, specify the possible USB VID/PID pair via **<USBID>** element with attributes "VID" and "PID". The attributes are hexadecimal values, and must be exactly 4 characters long. This is only needed for USB printers, not needed for remote devices or services.
    Following the example from Makerbot Replicator 2X profile, as these models may have boards with USB VID/PID of either 23C1/B015 or 23C1/B017.
    <USBIDs>
    <USBID VID='23C1' PID='B015'/>
    <USBID VID='23C1' PID='B017'/>

</USBIDs>

Printer Head Offset (mm) - defines the offset of additional heads from the first (id = 0)
print head.
<PrinterHeadOffset id='1' X='-29' Y='0' />

NozzleThickness (mm) - what is the physical diameter of the print head nozzle. (default is 0.35)
<NozzleThickness value='0.35' />

RealShapeOffset - the offset of the printed shape from the surface of the 3D model (should
be 0).
<RealShapeOffset value='0' />

Axial Homing Supported (bool) - does the printer support separate homing commands for each axis in
both directions (e.g. homing in both -X and +X). (default is false)
<AxialHomingSupported flag='0' />

Heated Platform Supported (bool) - does this printer have and support a heated platform/bed? (default is
false)
<HeatedPlatformSupported flag='1' />

Negative Coordinates Supported (bool) - does this printer firmware support negative coordinates?
(default is false)
<NegativeCoordinatesSupported flag='0' />

Anchor Length (mm) - how much material to extrude before starting the object. (default is 25)
<AnchorLength value='15' />

Filament Load Feedrate (mm/min) - the extruder feedrate while loading new filament. (default is 70)
<FilamentLoadFeedrate value='110' />

Filament Load Extrusion Length (mm) - how much to advance the extruder to load new filament.
(default is 50)
<FilamentLoadExtrusionLength value='80' />

Filament Unload Retraction Length (mm) - how much to reverse the extruder when unloading filament.
(default is 100)
<FilamentUnloadRetractionLength value='40' />

Needs ZIP (bool) - does the output file require ZIP compression before sending to the printer or service?
(default is false)

<NeedsZip flag='1'>

Full Color (bool) - does this printer support full color output? (default is false)
<FullColor flag='1'>

Coordinate System Centered (bool) - is the coordinate system of this printer centered? (default is false)
<CoordSystemCentered flag='1'>

Set Origin Supported (bool) - is the set origin command supported by this printer? (default is true)
<SetOriginSupported flag='0'>

Reset Extrusion Length (bool) - Should the extrusion length be reset after each printed layer? This can reduce the size of the generated GCode files, but some printers might not work properly when this is enabled. (default is true)
<ResetExtrusionLength flag='1'>

Nozzle Wipe Supported (bool) - does this printer have a place to wipe the nozzles? (default is false)
<NozzleWipeSupported flag='1'>

Extruder Steps Per Millimeter (number) - the steps per millimeter on the extruder, for firmware versions that don't have this predefined.
<ExtruderStepsPerMM value = '2047.62'>

Extruder Steps Per Revolution (number) - the number of steps created for one revolution of the extruder. This is used for printers that do not support explicit extrusion values for move commands (e.g. 3D Touch).
<ExtruderStepsPerRev value='2400>

Extruders Wait For Heated Platform (bool) - should the code wait for the heated platform to get to temperature before heating up the extruders. (default is true)
<ExtrudersWaitForHeatedPlatform flag='0' />

Fans Per Head (number) - the number of fans per print head. (default is 1)
<FansPerHead value='2' />

Move Steps Per MM (number) - the number of stepper motor steps per millimeter, used for systems that do not have this value in firmware.
<MoveStepsPerMM X='88.573186' Y='88.573186' Z='400' />

Anchor Location (mm coordinates) - where the anchor should be deposited before printing. (defaults to 0,0,0)
<AnchorLocation X='-20' Y='0' Z='0' />

Use Anchor Location (bool) - should the explicit anchor location be used (defaults to true).

        `<UseAnchorLocation value='1'/>`

Anchor Location Above Empty Space (bool) - is the anchor location above empty space (true), or over part of the print bed? (defaults to false)
`<AnchorLocationAboveEmptySpace flag='1' />`

Anchor Wipe Location (mm coordinates) - the location where the print head should wipe after extruding an anchor.
`<AnchorWipeLocation X='0' Y='0' Z='0' />`

Anchor Length (mm) - how much material to extrude for an anchor. (default is 25)
`<AnchorLength value='25' />`

GCode Wait for correct temperature command (string) - the GCode command number
for this printer to wait for the correct temperature. This varies in some firmware versions.
Use value -1 if this command is not supported. (default is 109)
`<GCODE_WaitForCorrectTempCommandId value='109' />`

GCode Remove Redundant Info (bool) - should redundant information be removed from the generated GCode. (default is true)
`<GCODE_RemoveRedundantInfo flag='0' />`

GCode Explicit Extrusion Supported (bool) - is explicit extrusion supported by this printer (E value in move commands) (default is true)
`<GCODE_ExplicitExtrusionSupported flag = '0' />`

GCode Implict Retraction Supported (bool) - does this printer implicitly retract filament when the extruder motor is turned on/off? (default is false)
`<GCODE_ImplicitRetractionSupported flag = '1' />`

GCode Extruder Turn On Command ID (string) - the GCode extruder on command for this printer, if it differs from normal. Set value to -1 if unused. (default is -1)
`<GCODE_ExtruderTurnOnCommandId value='101' />`

GCode Extruders Turn Off Command ID (string) - the GCode extruder disable command for this printer, if it differs from normal. Set value to -1 if unused. (default is -1)
`<GCODE_ExtrudersTurnOffCommandId value='103' />`

GCode Set Extruder Speed Command (string) - the GCode command to set the extruder speed for this printer, if it differs from normal. (default is 108)
`<GCODE_SetExtruderSpeedCommandId value='108' />`

GCode Prime Exruder Command ID (string) - the GCode command to prime the extruder on this printer, if it differs from normal. Set value to -1 if unused. (default is -1)

<GCODE_PrimeExtruderCommandId value='551' />

GCode Reverse Extruder Command ID (string) - the GCode reverse extruder command for this printer, if it differs from normal. Set value to -1 if unused. (default is -1)
<GCODE_ReverseExtruderCommandId value='561' />

GCode Enable Auto Retraction Command ID (string) - the GCode enable auto retraction command for this printer, if it differs from normal. Set value to -1 if unused. (default is -1)
<GCODE_EnableAutoRetractionCommandId value='227' />

GCode Disable Auto Retration Command ID (string) - the GCode disable auto retraction command for this printer, if it differs from normal. Set value to -1 if unused. (default is -1)
<GCODE_DisableAutoRetractionCommandId value='228' />

GCode Nozzle Wipe Start Command ID (string) - the GCode nozzle wipe start for this printer. Set value to -1 if unused. (default is -1)
<GCODE_NozzleWipeStartCommandId value='542' />

GCode Nozzle Wipe End Command ID (string) - the GCode nozzle wipe end command for this printer. Set value to -1 if unused. (default is -1)
<GCODE_NozzleWipeEndCommandId value='543' />

GCode Set Absolute Extruder Mode Command ID (string) - the GCode set absolute extruder mode command for this printer. If not valid, set the value to -1. (default is 82)
<GCODE_SetAbsExtruderModeCommandId value='-1' />

GCode Disable Motors Command ID (string) - the GCode disable motors command for this printer, if it differs from normal. (default is 84)
    <GCODE_DisableMotorsCommandId value='18' />

GCode Set Platform Temperature Command ID (string) - the GCode set platform temperature command for this printer, if it differs from normal. (default is 140)
<GCODE_SetPlatformTempCommandId value='109' />

Swapped YZ (bool) - are the Y and Z axes swapped on this device? (not normally needed, default is false)
    <SwappedYZ value='1'/>

Printer Type (string) - is this an FDM printer, or other. (not normally needed, default is fdm)
    <PrinterType value='fdm'/>

Quality Keywords

Quality (string) - section defining a named print quality. Allowed values are "high", "medium" and "low".
<Quality value='high'>
...
</Quality>

Mesh Repair Resolution (number) - defines the precision of the mesh repair algorithm. The value defines the resolution of a volumetric grid where the mesh repair takes place. The resolution is equal to 2^(number+2). For example, by using value 7, the mesh repair runs on a grid with resolution equal to 512 in each axis. Higher values allow to preserve smaller details but they in general lead to slower performance.
<MeshRepairResolution value='7' />

Layer Thickness (mm) - the thickness of vertical layers to print at this quality. (default is 0.25)
<LayerThickness value='0.1' />

Extrusion Width (mm) - the measured extruded width of the filament for this quality. This will generally be larger than the nozzle thickness. (default is equal to the nozzle size)
<ExtrusionWidth value='0.5' />

Nozzle Relative Z Offset (fraction) - what fraction of the layer height should the print head be offset in the Z direction. Some fused filament printers need additional clearance for the head to deposit the layer cleanly. (default is 0.5)
<NozzleRelZOffset value='0.5' />

Extra Perimeters (number) - the number of extra perimeters to generate around the outer edges of the object. (default is 1)
<ExtraPerimeters value='2' />   or <ExtraShells value='2'/>

Maximum Horizontal Traversal Feedrate (mm/minute) - the maximum movement rate of the print head in X and Y. (default is 2500)
<MaxHorizontalTraversalFeedrate value='9000' />

Horizontal Traversal Acceleration Time (seconds) - how much time to take to accelerate the print head from minimal feedrate to its maximum when traversing in X and Y. (default is 0)
<HorizontalTraversalAccelerationTime value='0.0' />

Maximum Vertical Traversal Feedrate (mm/minute) - the maximum movement rate of the print head in the Z direction. (default is 1100)
<MaxVerticalTraversalFeedrate value='2000' />

Minimum Horizontal Feedrate (mm/minute) - the minimum movement rate of the print head in X and

Y. (default is 100)
<MinHorizontalFeedrate value='170' />

Outline Feedrate (mm/minute) - the extruder rate while putting down the exterior outline of the object. (default is 1230)
<OutlineFeedrate value='3000' />

Interior Outline Feedrate (mm/minute) - the extruder rate while putting down interior outlines of the object. (default is the outline feedrate)
<InteriorOutlineFeedrate value='3000' />

Solid Fill Feedrate (mm/minute) - the extruder rate while putting down solid fills. (default is 1230)
<SolidFillFeedrate value='4000' />

Lattice Fill Feedrate (mm/minute) - the extruder rate while putting down interior lattice fill (infill). (default is 1230)
<LatticeFillFeedrate value='6000' />

Raft First Layer Feedrate (mm/minute) - the extruder rate while putting down the first layer of the raft (if a raft is enabled). (default is the outline feedrate)
<RaftFirstLayerFeedrate value='3000' />

Raft Feedrate (mm/minute) - the extruder rate while putting down additional layers of the raft (if a raft is enabled). (default is the outline feedrate)
<RaftFeedrate value='4000' />

Retraction Feedrate (mm/minute) - the extruder rate when retracting filament. This is frequently slower than the normal extrusion rate. (default is 1700)
<RetractionFeedrate value='1260' />

Retraction Length (mm) - how far to retract the filament when retraction is needed. (default is 3)
<RetractionLength value='1.8' />

Extra Post Retraction Extruded Length (mm) - how much extra extrusion to add after doing a retraction (to account for drip, etc.). (default is 0)
<ExtraPostRetractionExtrudedLength value='0.01' />

Extra After Retraction Move Length (mm) - the length of the head movement over the printed filament after its filament was retracted (wipes some leftover filament, reducing oozing artifacts). (default is 10)
<ExtraAfterRetractionMoveLength value='7' />

Outline Overlap (mm) - how much perimeters should overlap to close completely. (default is 0)
<OutlineOverlap value='0.05' />

Solid Fill Overlap (mm) - how much solid fill lines should overlap to fill completely. (default is 0.05)
<SolidFillOverlap value='0.15' />

Solid Fill Outline Overlap (mm) - how much should solid fill overlap perimeters to close completely. (default is 0.05)
<SolidFillOutlineOverlap value='0.45' />

Extruder Startup Length (mm) - define the length over which the extruder accelerates when it starts printing a new shape. (default is 10)
<ExtruderStartupLength value='10' />

Extruder Startup Feedrate (mm/minute) - the initial feedrate used at the beginning of an accelerated move.
<ExtruderStartupFeedrate value='170' />

First Layer Relative Speed (fraction) - how much slower to print the first layer. This is frequently much slower than normal printing speed to make sure that the first layer sticks correctly to the print bed. (default is 1.0)
<FirstLayerRelativeSpeed value='0.50' />

First Layer Relative Extrusion (fraction) - how much relative extrusion to make for the first layer? If printing without a raft, this accounts for the fact that your are printing against the print bed instead of another layer of plastic. (default is 1.0)
<FirstLayerRelativeExtrusion value='0.75' />

Lattice Fill Pattern (string) - The infill pattern to use on the interior of objects. The value can be "Diagonal", "CrossDiagonal", and "Cross". (Default is Diagonal)
<LatticeFillPattern value='CrossDiagonal' />

Lattice Fill Ratio (ratio) - how much of the interior should be filled? Smaller ratios use less material, but result in weaker objects. Higher ratios use more material, and result in stronger objects. (default is 0.2)
<LatticeFillRatio value='0.1'/>

Fan Disabled for First Layers (bool) - should the fan be disabled for the first few layers? This is sometimes done to improve adhesion to the bed. (default is false)
<FanDisabledForFirstLayers value='1' />

Maximum Length for Non Retracted Traversal Between Supports (mm) - how far can the print head move between supports without retracting the filament? (default is 2)
<MaxLengthForNonretractedTraversalBetweenSupports value='10' />

Traversal Z Offset (mm) - how much to move the head away from the print when moving horizontally but not depositing material? This is to avoid hitting existing material, but may cause problems on some printers. (default is 0)

<TraversalZOffset value='0.35' />

Raft Layers (number) - how many layers of raft to print before starting the object, if a raft is being used. (default is 3)
<RaftLayers value='3'/>

Raft First Layer Fill (ratio) - how dense should the first layer of the raft be? (default is 0.25)
<RaftFirstLayerFill value='0.2' />

Raft Fill (ratio) - how dense should subsequent layers of the raft be? (default is 0.25)
<RaftFill value='0.4' />

Thin Component Outline Overlap (mm) - how much should the outlines overlap on very thin areas? (default is 0.5)
<ThinComponentOutlineOverlap value='0.4' />

Print Thin Components With One Line (bool) - should thin components be printed with a single line, even if the geometry is smaller than the thinnest printable line. (default is false)
<PrintThinComponentsWithOneLine flag='0' />

Build Cleaning Tower (bool) - should a a cleaning tower be included when printing with multiple heads? This adds an extra object to the print to clean off the print heads and reduce contamination between colors. (default is false)
<BuildCleaningTower flag='1' />

Support Keywords

Extra Support Shells (number) - how many extra perimeters to print for supports? (default is 2)
<ExtraSupportShells value='2' />

Reference Support Size (mm) - the thickness of the reference support (used for calibration to determine the thickness of the generated supports). (default is 0.6)
<ReferenceSupportSize value='1.75' />

Real Reference Support Size (mm) - the real thickness of the reference support (this should be the same as reference support size for all GCode based printers). (default is 1.5) ???? Why do we have both ????
<RealReferenceSupportSize value='1.75' />

Reference Support Height (mm) - the height of the reference support. This is used for calibration to determine the thickness of the generated supports. (default is 20)
<ReferenceSupportHeight value='20' />

Minimum Distance of Support to Object (mm) - how close can the supports come to the object? (default is 0.5)
<MinDistanceOfSupportToObject value='1.0' />

Minimum Support Size (mm) - the minimal thickness of the support (excluding connectors and non trunk branches that end in connectors). (default is 0.45)
<MinSupportSize value='1.3' />

Minimum Connected Child Support Size (mm) - the minimal thickness of a non trunk support that ends in a connector. (default is 0.45)
<MinConnectedChildSupportSize value='1.0' />

Top Connector Size (mm) - the thickness of the top connector that connects the support with the printed model. (default is 0.45)
<TopConnectorSize value='0.75' />

Solitary Top Connector Size (mm) - the thickness of the top connector in case the support doesn't have any other branches (i.e. the whole support is solitary). (default is 0.25)
<SolitaryTopConnectorSize value='0.75' />

Top Connector Height (mm) - height of the top connector (where the support joins the model). (default is 1.5)
<TopConnectorHeight value='0.75' />

Top Connector Shape (string) - cross-section of the top connector (right now the only allowed value is "Square")
<TopConnectorShape value='Square' />

Top connector Maximum Distance On Outline (mm) - the desired spacing of support connectors on a single perimeter. Larger values result in sparser supports. (default is 5)
<TopConnectorMaxDistanceOnOutline value=" 5"/>

Top connector Maximum Eccentricity On Outline (mm) - the desired maximum distance of an unsupported point of a single perimeter from a line connecting two nearest supported points. (default is 0.5)
<TopConnectorMaxEccentricityOnOutline value=" 0.5"/>

Create End Support Contact Points (bool) - should the supports make connections between trunks and its branches. (default is true)
<CreateEndSupportContactPoints flag='1' />

Create Inter Support Connections (bool) - connect disjoint support trees with slabs to increase stability and reduce extruder jumps. (default is false)
<CreateInterSupportConnections flag='0' />

Dissolvable Top Connector Size (mm) - the thickness of the top connector when it is printed with a dissolvable material. (default is top connector size)
<DissolvableTopConnectorSize value='2.2' />

Minimum Slope (degrees) - the minimum slope that requires supports. (default is 30)
<MinSlope value='30' />

Additional Slopes Data - we allow to define alternative support parameters fro different values of slope angle. This feature can be used to generate supports even on surfaces that have higher slope than the MinSlope defined above. Usually these supports are sparser as they use larger spacing. The format for extra slopes is shown below. In the current version we support only one extra slope.
<ExtraSlopeData>
        <MinSlope value='45.0' />
        <TopConnectorMaxDistanceOnOutline value='25.0' />
        <TopConnectorMaxEccentricityOnOutline value='7' />
</ExtraSlopeData>

Material Keywords

Material Name (string) - the name of the material to be displayed in the application UI.
<Material name='PLA'>

Material Type (string) - the actual type of material. Currently supported values are "ABS", "PLA", and "PVA". (default is PLA)
<MaterialType value='PLA' />

Needs Raft (bool) - does this material need a raft or not. This applies primarily to fused filament printers. (default is false)
<NeedsRaft flag='0' />

Needs Supports (bool) - does this material need supports generated. Most FFF printers will need this enabled. (default is false)
<NeedsSupports flag='1' />

Needs Escape Holes (bool) - should escape holes be generated for unused print material to escape the interior of the model. This is primarily for powder fusion printers. (default is false)
<NeedsEscapeHoles flag='0' />

Head Temperature (degrees C) - the temperature of the head to print this material. (default is 215)
<HeadTemperature value='200' />

MaxPlatformTemperature (degrees C) - the temperature of the bed for this material. (default is 0)

```
<MaxPlatformTemperature value='60' />
```

Filament Diameter (mm) - the average diameter of this type of filament. (default is 1.75)
```
<FilamentDiameter value='2.87' />
```

Extrusion Multiplier (ratio) - a multiplier to fine tune the filament extrusion rate. (default is 1.0)
```
<ExtrusionMultiplier value='1.0' />
```

Time to Solidify (seconds) - how long this material takes to solidify before additional material can be deposited on top of it. (default is 5)
```
<TimeToSolidify value='15' />
```

Minimum Wall Thickness (mm) - the minimum thickness for walls and thin components. (default is 0)
```
<MimimumWallThickness value='2'>
```

Maximum Inactive Time (seconds) - how long can the head be left inactive? (default is 240)
```
<MaxInactiveTime value='2 '/>
```

Inactive Filament Retraction Length (mm) - how much to retract the filament if the head will be inactive. (default is 1.5)
```
<InactiveFilamentRetractionLength value='2' />
```

Dissolvable (bool) - is this a dissolvable support material? (default is false)
```
<Dissolvable value='1' />
```

Tool Change Filament Extruded Length (mm) - how much to extrude after changing heads. (default is -1, or unused)
```
<ToolChangeFilamentExtrudedLength value='2' />
```


Speed Multiplier (ratio) - Speed multiplier for this material. (default is 1.0, normally not needed)
```
<SpeedMultiplier value='1'/>
```

Price Per Cubic Centimeter (number) - the price (in US dollars) per volume for this material. This is used in estimating costs for online services.
```
<PricePerCubicCM value='2'>
```

Price Per Square Centimeter of Surface (number) - the price (in US dollars) per surface area for this material. This is used in estimating costs for online services, for certain processes that have a surface area pricing.
```
<PricePerSquareCMSurface value='0.5'>
```

Handling (number) - the price (in US dollars) for handling when printing using an online service. ????
This should have a more descriptive name, maybe "service fee" "service baseline" "service price" ????

<Handling value='3'>

Diffuse Color (RGB float) - the diffuse color to preview this material in the UI. (default 0.2,0.2,0.2)
<Diffuse R='0.8' G='0.3' B='0.3' />

Specular (RGB float) - the specular color to preview this material in the UI. (default 0.8,0.8,0.8)
<Specular R='0.6' G='0.6' B='0.6' />

Illumination (RGB float) - the illumination to preview this material in the UI. (default 0,0,0)
<Illumination R='0' G='0' B='0' />

Ambient (RGB float) - the ambient color to preview this material in the UI. (default 0,0,0)
<Ambient R='0' G='0' B='0' />

Shine(float) - the shine factor to preview this material in the UI. (default is 0.2)
<Shine value='0.8' />

Reflection (float) - the reflection factor to preview this material in the UI. (default is 0)
<Reflection value='0' />

Reflection (float) - the reflection factor to preview this material in the UI. (default is 0)
<Roughness value='0' />

Bump (float) - the bump depth factor to preview this material in the UI. (default is 1.0)
<Bump value='0.1' />

Opacity (float) - the opacity to preview this material in the UI. (default is 1.0)
<Opacity value='1' />

Refraction (float) - the index of refraction to preview this material in the UI. (default is 1.0)
<Refraction value='1.46' />

Plate Levelling Keywords
All of the Print Plate Leveling settings are contained within the <PrintPlateLeveling>
node.
For the positioning values, we use the Normalized Coordinate System, illustrated at
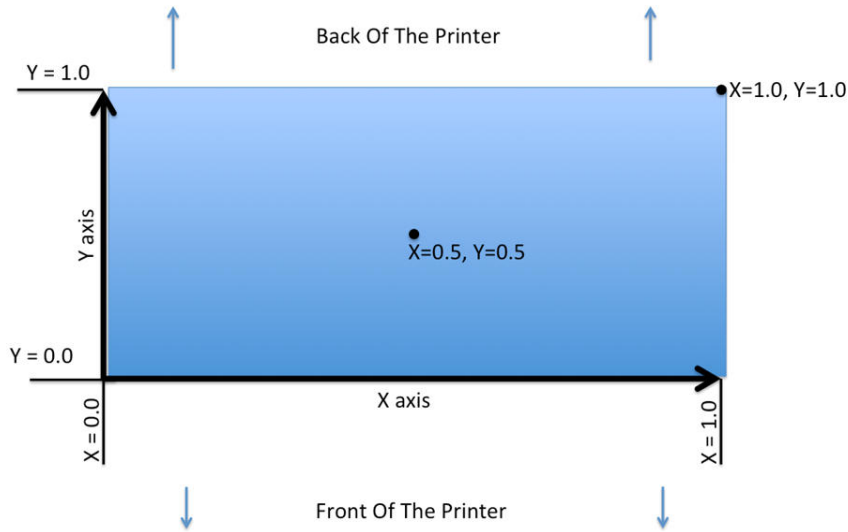Figure 1.

Figure 1: Normalized Coordinate System of a 3D Printer Plate, view from top.

In the Dialog, the displayed points are called by their descriptive location (e.g. "Top Left"), and are named and indexed as illustrated in Figure 2. Those names are fixed and cannot be customized in the profile due to Photoshop localization.
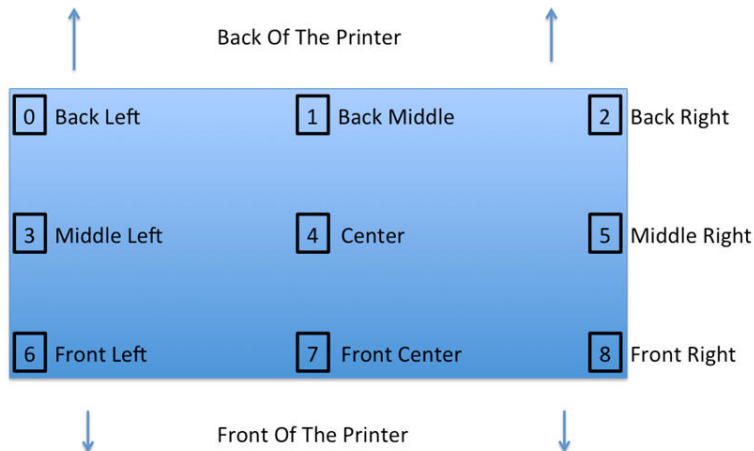

Figure 2: DisplayedPointIndexes and names of a 3D Print Plate Points, view from top.

Normalize Home Position (XY ratio) - the normalized position (see Figure 1) of the head on the bed in the home position. The DisplayPointIndex is the index of the displayed point in the dialog that is closest to the printhead "home" position, per convention displayed in Figure 2.(0 is farther left corner, 2 is farther right corner, 8 is near right corner, etc.)

<NormalizedHomePosition X='1.0' Y='1.0' DisplayPointIndex ='2' />

<u>Rivets (XY ratio)</u> - the normalized head positions (see Figure 1) of the control screws for levelling the bed. Photoshop supports up to 8 rivets.
<Rivet0 X='1.0' Y='0.0' />
<Rivet1 X='1.0' Y='1.0' />
<Rivet2 X='0.0' Y='0.50' />

<u>Checkpoints (XY ratio plus rivet number reference)</u> - normalized head positions and order in which test them for print bed levelling, in the Photoshop's Print Leveling Wizard dialog. Each checkpoint has the following attributes:

      X: Normalized X coordinate (see Figure 1)
      Y: Normalized Y coordinate (see Figure 1)
      ControlRivet: Index in the rivets array for the corresponding rivet.
      DisplayPointIndex: The index of the displayed point in the dialog that is closest to the checkpoint,per convention displayed in Figure 2.(0 is farther left corner, 2 is farther right corner, 8 is near right corner, etc.)

The final test is indicated by setting the ControlRivet value to 100. Photoshop supports up to 16 check points.

        <Checkpoint0 X='0.5' Y='1.0' ControlRivet='0' DisplayPointIndex='1' />
        <Checkpoint1 X='0.8' Y='0.0' ControlRivet='1' DisplayPointIndex='8' />
        <Checkpoint2 X='0.2' Y='0.0' ControlRivet='2' DisplayPointIndex='6' />
        <Checkpoint3 X='0.5' Y='1.0' ControlRivet='0' DisplayPointIndex='1' />
        <Checkpoint4 X='0.95' Y='0.0' ControlRivet='1' DisplayPointIndex='8' />
        <Checkpoint5 X='0.05' Y='0.0' ControlRivet='2' DisplayPointIndex='6' />
        <Checkpoint6 X='0.5' Y='0.5' ControlRivet='100' DisplayPointIndex='4' />

Suggested approach for profiling a new printer

Find an existing description for a printer similar to yours, duplicate the file and change the name. Set the printer location and file format that it supports, then the printer size, area, resolution, nozzle size, and number of print heads. Most of these values can be copied from Slicer, Repetier or similar application configurations. If your printer is a local USB device, don't worry about the firmware signature yet - you can still generate gcode without a direct connection to the printer. Then change one of the existing materials to match what you are using. Double check the filament diameter and head temperature!

Now change the high quality setting to 0.3 mm -- because you're going to want to print some tests quickly, and 0.1 mm will take longer. Make some small test prints by exporting GCode and printing via Reptier, Pronterface, or similar printer apps -- this gives you a chance to double check the GCode and see

more of what is going on (or wrong) with the print. We suggest printing something like a 2 cm box (you can get lots of small test shapes from Thingiverse), or a very low extrusion shape made in Photoshop (remember, you may have to print several of these). Fix the parameters that aren't right, and print again until results are reasonable.

   Change the quality setting to 0.1 mm, change the extrusion rates, perimeter counts, shell counts, etc. to what you need for your printer (some of these values can be copied from Slicer or similar apps). Make another test print, fix the parameters that aren't right, iterate.

   If the prints are working so far, then all that's left is fine tuning the print parameters, and possibly defining the USB firmware signature string or USB vendor/product codes. Now you can add other materials, change the bed levelling parameters to match your printer, print with supports and fine tune those parameters, etc.

Adjusting mesh repair quality

The quality of models generated by our mesh repair algorithm is affected two different parameters: printer resolution and mesh repair resolution (see xml snippet below):

```
<Printer>
    <PrinterResolution X='10' Y='10' Z='10'/>
    <Quality value='high'>
        <MeshRepairResolution value='10' />
    </Quality>
    <Quality value='medium'>
        <MeshRepairResolution value='9' />
    </Quality>
</Printer>
```

"PrinterResolution" defines desired mesh repair sampling per one mm. The inverse of printer resolution is the minimal feature size. For example for printer resolution 10 the desired minimum feature size is $1/10=0.1$mm. Any features on the input model smaller than this value are most likely going to disappear during the mesh repair process. The second parameter that can affect the minimum feature size is MeshRepairResolution, which is used to compute the maximum allowed number of samples per axis. For <MeshRepairResolution value='R'/> the maximum number of samples per axis is $2^{(R+2)}$. For example when R=7, the maximum number of samples is 512. The maximum number of samples is used to clamp the value computed from the "PrinterResolution" parameter. The main purpose of this parameter is to limit memory consumption and improve performance of the mesh repair algorithm.

If your goal is to improve the quality of models processed by the mesh repair algorithm, it is important to first determine which parameter is limiting the number of used samples. Often, for small scale models, the PrinterResolution needs to be increased to properly capture all details. On the other hand, large scale models are usually limited by the MeshRepairResolution parameter. Note that increasing the mesh repair resolution above 9 can result in a large memory consumption and very long mesh repair processing times.

Definitions of Terms / Glossary

Anchor - material extruded before starting the object. This is to clear old material from the head, replace material that may have dripped out, and get the head ready to print.

Filament - the raw material fed into the 3D printer, frequently a round filament of plastic. In some printers it may actually be a solid or liquid.

Fused Filament Fabrication - the most common form of 3D printing that uses a heated plastic filament deposited in layers to build up objects.

GCode - a simple text based language specification for instructions sent to many CNC machines and 3D printers. There are variations in the GCode instructions required by different printer models. And some printers use proprietary binary variants of GCode.

Head - the part of the printer that deposits or removes material to make the solid object.

Infill - how the interior of the printed object is filled. This can be varied to reduce material use, reduce print time, increase strength, or intentionally leave an object hollow.

Perimeter - the outline or outer edges of the object on each layer.

Powder Fusion - a method of 3D printing that binds layers of powder (plastic, metal, or gypsum) to create the 3D object.

Raft - a few layers of loosely connected material deposited before the object to improve adhesion on the bed, account for irregularities on the bed, and annoy professionals who are experienced at 3D printing.

Solid Fill - solid areas printed near the perimeters to improve strength, and on the top and bottom surfaces of the object.

Shells - also called Horizontal Shells - the upper and lower surfaces of the objects, printed solid. If you can see the infill through the top or bottom of the object, you don't have enough horizontal shells.

Support - extra geometry added to support overhangs and shapes that are not normally printable with some types of 3D printing. The supports are designed to be easily removed from the final print. These may be printed in the same material as the object, or in another material designed to be easy to remove.

Photoshop is a registered trademark of Adobe Systems Incorporated.
All other trademarks referenced in this document are property of their respective owners.