

A Guide to Clay 3D Printing

By Jonathan Keep (November 2020)

A Guide to Clay 3D Printing	1
Clay 3D Printing	5
Limitations	5
Printers	6
The Rig	6
Printhead	6
Delivery	7
Compressed Air or Mechanical Ram Delivery	7
Slice Layer	8
Nozzles	8
Visualisation of Nozzle size to Wall thickness to Layer height	9
Setting Start Z Height	10
Boards and Bats	10
Print Bases	10
Infill	10
Support	11
Drying	11
3DFiles for Clay Printing	12
Generating 3D Content	12
3D Drawing	12
Scanning	13
File Download	13
Coding	14
Augmented Reality	14
Hints and Tips on Preparing 3D Files	15
Thickness	15
File size	15
Watertight file	15
Surface norms	15
Added Support	15
File Types	15
Slicing	16
Clay for 3D Printing	17
Clay Types	17
Additives	18
Clay Consistency	18
Hard or Soft Clay	19
Mix from Wet or Dry	19
Coloring clay	19
Clay Shrinkage	20
Softening a bag of throwing/modeling clay by hand	20
Mechanic clay preparation	20
Clays known to 3D print well	21
Porcelain	21
Stoneware Clay Recommendations	21

	2
3D Printing Clay recipe	22
JK Print Mix	22
Convert an AnyCubic Kossel for Clay 3D Printing	23
Rig - The AnyCubic Kossel	24
Clay Printhead	26
Clay Delivery	27
Self Build Clay Printhead	31
The Printhead	31
Parts	31
Feed pipe	33
Nozzles	34
Screws	34
Cura for Clay 3D Printing	36
Introduction	36
Layout	36
Setting up Printer Profile	37
Printhead Settings	38
Extruder Settings	38
Setting up Material Profile	39
Setting up Print Profile	39
Quality	39
Shell	40
Infill	40
Material	40
Speed	41
Travel	41
Build Plate Adhesion	41
Mesh Fixes	41
Special Modes	41
Experimental	42
Import Export Print Profiles	42
Cura Workflow	43
Model Editing Tools	43
Right Click Model Menu	43
Slice Button	43
Slice Preview	43
Save gcode	44
Appendix - Detailed Research Documentation	44
1 - Testing of the relationship between Nozzle Size, Layer Height, Print Speed and Material Consistency for Clay Extrusion 3D Printing	44
Introduction	44
Benchmark Print	44
Nozzle to Layer Height Tests	45
Procedure	46
Vertical Edge Test	47
Low relief Pattern Test	48
Overhang Test	49
Speed Test	50
Stiff to Hard Clay Test	51
Conclusions	53
Nozzle to layer height proportion	54

	3
Print speed	54
Stiff to Hard Clay	54
2 - Testing of Six Clays for Extrusion 3D Printing	55
Introduction	55
Clays Tested	56
Clay Paste Consistency	56
Test Procedure and Equipment	56
Water Ratio	57
Clay Consistency - Drop Spike	57
Syringe Pressure Extrusion	57
Tube Pressure and Flow Rate	57
3D Print Quality	58
Test Results	58
Clay Shrinkage	61
3 - Formulating and Testing a Clay Body for Extrusion Clay 3D Printing	64
Introduction	64
The Clay Recipe	64
Test Mix 1	64
Water to Dry Ingredients Proportions	65
Measurements of Clay Consistency	65
Printing Tests	66
Cylinders	66
Edge, Texture, Overhang Test Shape	67
Additives to Improve Plasticity and Elasticity	68
Deflocculation	68
Paper Fiber	68
Bentonite Plasticiser	69
Reformulated Receipt: Test Mix 2	69
Test Mix 2	69
Test Clay Mix Comparisons	70
4 - Testing Fine Porcelain Extrusion Printing	70
Introduction	70
Clay Samples and Clay Preparation	71
Specific Gravity Measurement Tool	72
Equipment	73
Process	75
Conclusion	78
5 - JK Drop Spike - Clay Consistency Measuring Tool	78

Clay 3D Printing



I often get asked by people interested in ceramic as to whether I think 3D printing will be the future. My answer is no, it is an add on. I am a traditionalist, I have a traditional ceramics background, but then I see tradition as a continuum. In time 3D printing will just become considered part of the ceramic tradition. It is a forming process that will become added to all the other ways there are to work in clay.

There is a steep learning curve to ceramic 3D printing. Not only is there all the computer skills and the machine technical knowledge to be learned but then there is all the ceramic knowledge too. It is a field for collaboration and sharing of skills. There are different approaches, those who are more technical and their interest will be in the engineering, and then there are those who are more creative and will probably break all the rules to see what can be done with the process.

This helps illustrate how there is no one way of working in ceramic 3D printing. That goes for the equipment that is used and also for the style of working. For more architectural forms one kind of machine might work best while for more organic forms another. Some people are interested in the patterns the clay extrusion makes while others only in the form. I am a great believer in keeping things as simple as possible and to be comfortable with what works for you. This is a guide and in no way represents the only way to work in clay 3D printing.

Limitations

One thing is for sure extrusion clay printing has its limitations. It is vertically oriented, by this I mean it is very good at piling up but no good at making flat curved things. It is relatively slow and not very precise. The machine might be precise but the clay moves around a lot. If you want to produce lots of the same thing then traditional ways of working like moulding is still best. What 3D printing is good for is making individual or customised objects. There is a certain irony that mechanisation is often thought of as mass production but here we have a process where each time you run the

machine you may as well run a different file. Printing offers the possibility of making numbers of unique and distinctive objects that are so often associated with the hand made.

Printers

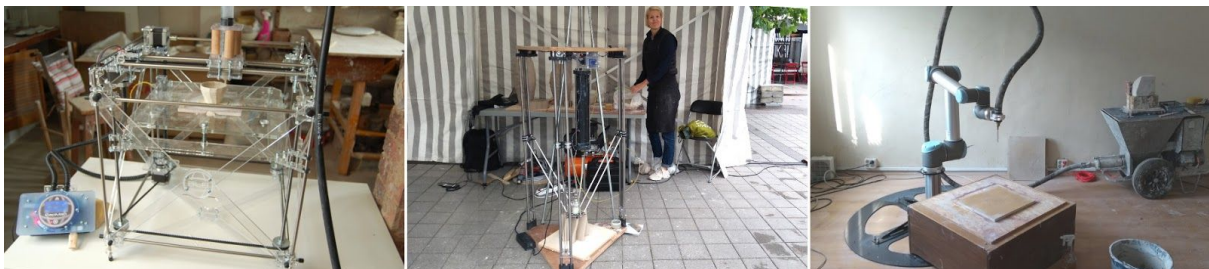
When thinking about clay extrusion type printers I like to break them up into rig, printhead and clay delivery. By the rig I mean the computer controlled machine that will move the printhead in 3D space. The printhead is the mechanism that deposits the clay layer by layer and the delivery is the process of getting the bulk clay material to the printhead.

The Rig

There is no perfect machine and different ways of working will suit different configurations. Budget also becomes a consideration as a six axis robot arm can be used as a printing rig but will not necessarily fit the budget. Conventional 3D printing is about piling up two dimensional sliced profiles so unless you are using all the axes of a robot arm there are cheaper options.

Leaving aside using robot arms for 3D printing, you are more than likely looking at the standard cartesian box type configuration or the delta style of rig. My first printer was a RapMan, copied from Belgium design studio Unfold and that was a box, standard cartesian design. Then, when 3D Systems closed down the production of the RapMan and I was looking to build my own machine, my preference was for a Delta style. Not being an engineer the Delta looked easier to build and I did like that the base and print remained in one place. I make organic shapes that want to fall over during printing so I do not want a build plate that is moving. I also liked the way the delta arms move and the movement is very smooth.

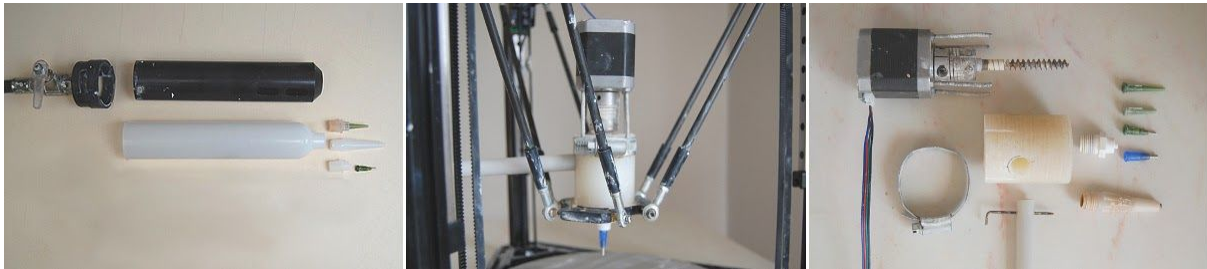
In comparison and in general terms the cartesian design can carry more weight. The clay container (delivery) can be closer to, or be part of the printhead. This configuration is also better if you need width and depth, more horizontal distance. However with some designs it is not uncommon for the printhead to move during printing, so be assured that your style of work will suit this action. Also, that if you plan to print complicated shapes at quick speeds that the movement can cope with this and does not judder itself out of alignment.



Printhead

My original JK Self Build Delta had a continuous flow printhead. You manually switched on the flow and the clay kept coming until you switched it off. The flow was regulated by air pressure and the clay consistency had to be uniform to ensure even printing. The development of the screw or auger printhead, that stops and starts the extrusion and is controlled by the gcode from the slicing software was a huge step forward. It offered the possibility of making much more complex forms while also regulating the flow of material so much better.

A screw/auger printhead is relatively easy to implement as it makes use of the gcode feed rate settings that are standard to plastic 3D printing. On a plastic printer a motor is controlled to push the plastic filament into the printhead at a given rate. On a clay printer the motor turns a screw that pushes clay at a given rate. The motors are the same but the rate of flow differs and these setting adjustments are made in the slicing software. The screw or auger is nothing special and can be adapted from parts bought at a hardware shop. What is more difficult is devising a way of getting a constant flow of clay to the printhead and this is why I like to consider 'delivery' as a separate issue.



Delivery

Returning to my first Delt design, the clay delivery was from a pressurised glue gun type cartridge carried on the printhead. This was simple and worked well but there is a limit to the size and weight that can be carried on a moving printhead. Workflow is important to me and needing to constantly change and fill small clay cartridges lacked appeal. Changing to a large clay container of 3 litres that sits alongside the printer delivering clay for a day or two of printing was just such a pleasure in comparison.

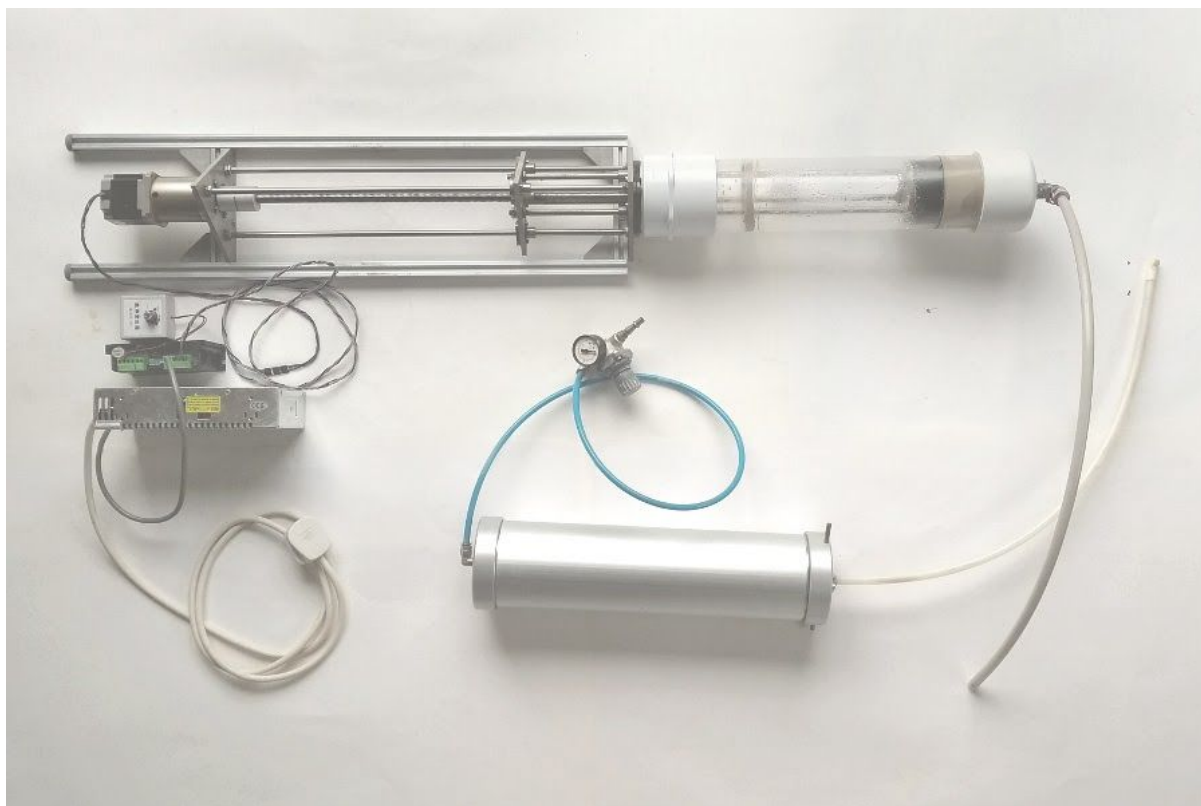
When considering clay delivery bare in mind that as you increase the scale of a three dimensions object the amount of material needed increases exponentially. If you double the size of a 10 cm object, the wall will need to become thicker and the volume and therefore the surface increases. A 20 cm object of the same proportion will need eight times as much material! Hint, make sure you have a good volume for your clay delivery.

Compressed Air or Mechanical Ram Delivery

Personally I prefer compressed air. I have both systems and both work equally well. What I like about compressed air is that once you get the clay flowing, and for me this is between 4 - 6 bar of pressure you can just leave it. If there is a blockage the pressure just holds and nothing breaks. If there is a problem with clay flow and you turn up the air pressure, it is immediately. With compressed air there are also not more slicing settings to consider. Most of all an air system is simple and easier to clean and maintain. I am always concerned that something is going to break on the mechanic ram. The one drawback is that you need an air compressor but I have that as I use it for spraying glaze.

There is the case for mechanical rams being able to push stiffer clay. This could come down to the personal style of work and design of printer. I have found no advantage in using clay any stiffer than I can push down a 70 cm long, 10 mm inside diameter nylon tube at 4 to 5 bar of air pressure. Clay under pressure dewateres and the higher the pressure, ram or air, the more the water gets forced out first. The greater pressure required to push harder clay just results in more compaction of the clay in your ram towards the end of the container. Also I have found using harder clay on curved forms can

result in delayering or cracking. My approach is to err on the side of soft clay and use heat lamps or warm air to stiffen the clay as I print.



Top, mechanical ram with electronic control, left. Lower right compressed air clay delivery. These two pieces of equipment offer the same amount or volume of clay delivery.

Slice Layer

The slice or layer height is dependent on the wall width and personal preference. As a general guide the slice height is one third of the wall width. The flatter your layer height in proportion to wall width the more stable will be your print especially when the wall begins to build out or in. If you prefer a more rounded look to the print layers then the height to width will be more of a proposition of one to two. However do make sure your layers are well compressed together or you can get delayering during drying..

The wall width is dependent on the nozzle size. The clay extrudes a little wider than the nozzle. So to put some numbers to this. When using a 1.6 mm nozzle the wall spreads to 1.8 mm / 2 mm and I set the slice height in the slicing program to 0.6 mm. A 2 mm nozzle results in a wall width of around 2.2 to 2.5 mm and the slice height is set to 0.8 mm. On a larger machine using a 3.5 mm nozzle the print wall is about 5 mm / 6mm wide and I use a slice height of 1.2 mm. So here the slice height is more like one quarter of the wall width.

Nozzles

The nozzle size changes with the scale of the print. The nozzle size will also determine the amount of detail that can be obtained. A corner or edge can only be as sharp as the diameter of the clay extrusion. Also as discussed above the nozzle influences the layer height so if you are looking for a fine layer height you will probably use a small nozzle.

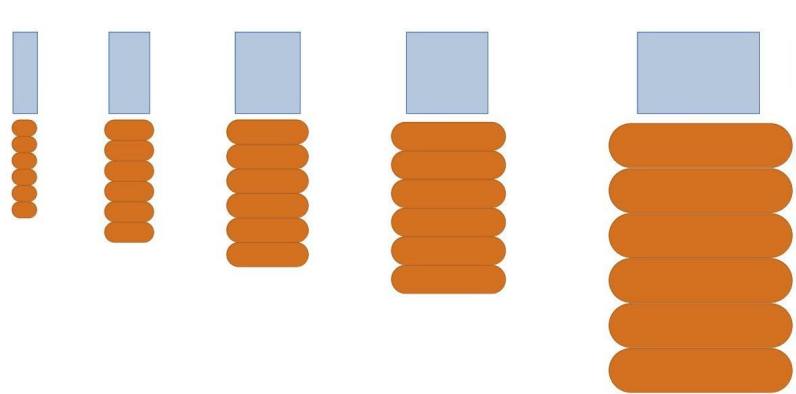
If I am looking for detail in a print that is 20 to 30 cm high, I will use a small nozzle of 1.6 mm but I will slice my model to have a double thickness wall. The layer height will be 0.6 mm. This will be slow to print but gives strength, stability and fine detail to the wall.

A 2 mm nozzle is probably a good starting size for printing 10 to 20 cm tall objects with a single wall. You can obviously use any size nozzle that you want but if you are looking for a large extrusion then firstly make sure your machine can push through enough clay for the size of the nozzle. Secondly as I indicated when talking about clay delivery, as you scale up the size of your nozzle the amount of material you will use increases hugely.

There is a lot more information on nozzle size and layer height in the appendix [1 - Testing of the relationship between Nozzle Size, Layer Height, Print Speed and Material Consistency for Clay Extrusion 3D Printing](#).

Visualisation of Nozzle size to Wall thickness to Layer height

Below is a quick sketch that is roughly in proportion offering a visualisation of nozzle size (blue grey) to extrusion width and height. (Six layer extrusion) Obviously there is no ideal here and this represents my interests but can be presented as an approach. This is looking for the least obvious layering pattern with the most horizontal reach, so stable cantilevered or curved single skin walls can be printed. My observation is that as the nozzle gets bigger there is more spread of the extrusion so the wall is that much wider than the nozzle. Hence considering the ratio of layer height to wall thickness rather than layer height to nozzle size.

Nozzle Size mm	0.6	1	1.6	2	4
Extrusion width or wall thickness	0.6	1.1	1.8	3	6
Layer height	0.4	0.5	0.6	0.7	1.2
Ratio of layer height to wall thickness	2:3	1:2	1:3	1:4	1:5
					
All measurements in mm Ratios rounded off					

Setting Start Z Height

The start z height or what is often called leveling in plastic printing, varies from machine to machine. In the most basic printers such as the JK Self Build delta the z height is set in the printers firmware. Each time you run the printer it starts printing at the same height. The simplest way to make a change to the height, if for example you change the length of the nozzle is to change the thickness of the boards underneath the start point.

Other machines have a height sensor that is attached to the printhead each time you want to reset the start height from the default firmware setting. The new start height is retained until you either revert back to the default firmware settings or reset the height with the sensor. On other machines the start height can be set or altered in the settings on the lcd screen of the control panel. If this is the case then under the 'Prepare' option on the lcd screen you will find something to do with adjusting the z height. Whatever the system it is important that the printhead nozzle is just clear of your print bed so that the first layer of your print squished down and attaches to whatever you are printing onto. A good clearance is just under half your slice or layer height.

Boards and Bats

It is preferable to print onto a board or bat so completed prints can be easily removed from the machine. As it is important to get the first layer z height correct on each print it makes sense to standardise the thickness of all your boards.

Boards or bats are best made from a porous material that is as smooth as possible so that the wet clay dries away from the board. Plaster of paris bats or unsealed wooden boards are good. Wet the board or plaster with a damp sponge just before printing to ensure that the clay extrusion initially sticks to the removable surface.

Print Bases

If you have an object that requires a clay base this can be printed. You do not draw a thickness to the base in your 3D model but set it up in the slicing program. There will be a setting for top and bottom thickness. In the Cura software you can select just a bottom thickness. The thickness will be printed in layers and I would suggest to print at least three layers. So the thickness you should enter is three times your slice or layer height. However be aware that printed bases, particularly as they get bigger can have a tendency to crack. This problem can be reduced by printing onto a porous plaster of paris bat so that the wet clay does not stick as it dries.

My preference if printing objects with a large base is to roll out a slab of clay and print onto the clay. The clay slab can be quite stiff and your z height setting does not have to be so precious as it is not a problem if the nozzle cuts into the clay slab a little. In the slicing program you prepare your model to be a wall only. After printing excess clay from the base is trimmed away from around the print once it is stiff enough to handle without distortion.

Infill

The infill settings available in slicing software can be used for clay printing but tends to become quite messy until you refine all your settings. The slicing programs offer many settings that will need

experimenting with. If you are considering infill it is advisable to print two or more walls so that the outer wall prints clean and only the inner wall is broken by the infill pattern.

Support

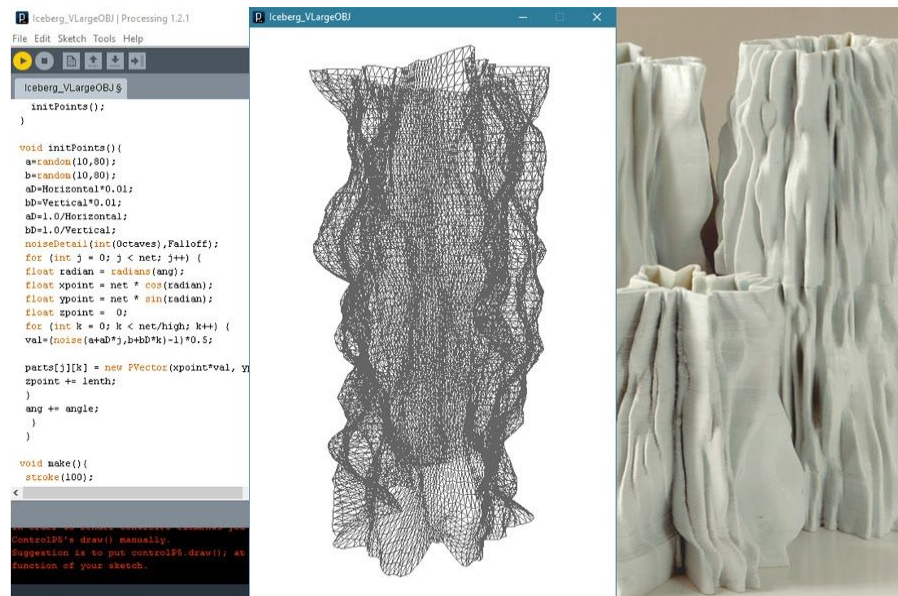
Most support offered by slicing software is too flimsy to be built in soft clay. The work around is to build solid printable support into your model at the drawing stage before slicing. Alternatively while printing and by hand you can shape support in clay and offer it up to the print as required. Short lengths of fine rigid wire can be useful to lay across the print as it builds to give support to awkward areas. Wire bracing laid across a form can help give rigidity to a shape while it is being printed. These lengths of wire are withdrawn after printing and once the form is stiff but before it becomes bone dry. The small hole in the surface can be touched up with a bit of clay.

Drying

As clay dries it shrinks so uneven drying should be avoided as this can create distortion and tension that at worst can result in cracking. Objects should be allowed to dry evenly. Most prints can be left to dry naturally in a room but if there is infill or there are internal structures you might throw some plastic over it to dry more slowly. Clay objects must be bone dry before being fired in a kiln.

Prints can be dried while printing to stiffen and become self supporting so as not to collapse. If your shape is self supporting then it is best to avoid forced drying. If you do dry do not dry too strongly. Preferably dry all around your print and once you start drying it is best to keep drying. This is not essential but drying ever so slightly changes the character of the surface and on a pure form any on off of drying becomes visible as a distortion of the surface. Heat lamps (as for heating pet amphibians) or photographic lamps are good as the radiant heat is not too strong but must be moved around the form. Warm air is good for drying but the air movement must not be so forceful that it moves the print wall.

3DFiles for Clay Printing



It is going to be your 3D files that you prepare for ceramic printing that will make you stand out from the crowd. Anybody in the crowd will probably be able to work a clay printer but it will be your creativity and individuality that will make what you print special. Well made files print better and there are any number of ways and combinations of techniques available to generate digital 3D content.

Generating 3D Content

3D Drawing

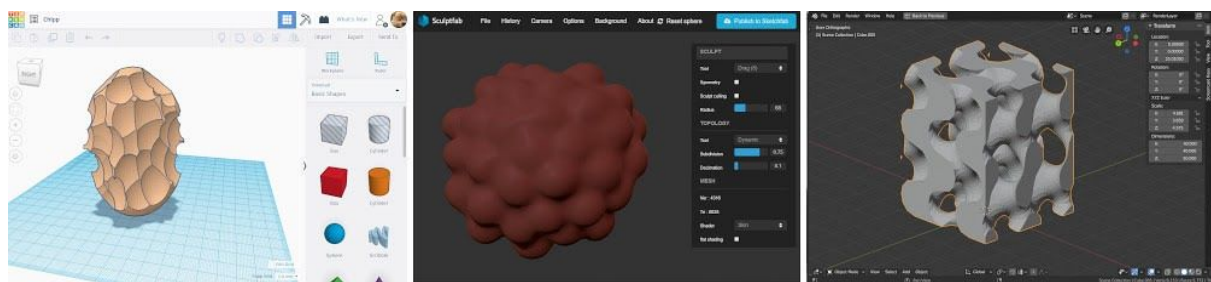
To make a start you do not need to be daunted by high end specialist 3D modeling software. There are two easy to use free web-based solutions that first introduces building with primitive shapes and secondly a sculpting program that is like working in virtual clay.

Tinkercad <https://www.tinkercad.com/> might look like a kids toy but give it time and the possibilities are endless. The principle is to stick or boolean forms together or to subtract, cut away shapes. Ensure that before you download a.stl file for printing that you have grouped all your shapes, positive and negative together into one watertight shape. This ensures there are no ungrouped intersection surfaces or elements floating inside your model as these can cause problems when you come to slice, or prepare the .gcode for your printer. Note that 3D files can be imported in Tinkercad so this program can be used for editing, distorting, cutting up, mashing together 3D forms found or generated from other sources.

Sculptfab <https://labs.sketchfab.com/sculptfab/> is a much more organic drawing tool. With a selection of virtual brushes you sculpt onto a sphere. Files can be imported into and

exported from Sculptfab so once again it can be used in conjunction with other software. Sculpted 3D meshes can be very detailed resulting in large file sizes so if this is the way you want to work, get to understand the topology settings, where the mesh detail can be reduced with the decimation slider.

Blender <https://www.blender.org/> is my choice of 3D drawing program when you are ready for a high end modeling software. Developed for animation rather than engineering it is not everybody's first choice but being open source, free to download and use, this incredibly powerful software never ceases to amaze me. There are no shortcuts to learning how to use it, you just have to put in the hours. With a huge community following you will never be short of online tutorials to help you along. Blender offers mesh modeling like Tinkercad but also Sculpting like Sculptfab in the one platform, and much, much more. It is not bad as a video editing program either, if you need one and it is all free.



Scanning

3D scanning can give you a printable digital file taken from physical objects. Having kept an eye on accessible 3D scanning for some years the techniques of photogrammetry are offering some of the most interesting results. This is where a number, 30 to 60 photos are taken from different angles of the object to be copied and then the program stitches this 2D information into a 3D mesh.

Agisoft <https://www.agisoft.com/> is a software that I have used and produces good results. It is proprietary software but if you search out a 30 day trial you can try out it's capabilities.



File Download

Copy, cut and paste, or the mash up, offers possibilities that reflects contemporary digital ways of working. There are masses of 3D files for download on the internet but I would hope your interests in 3D printing are to make your own original works. As in music remixes or graphic collages, file downloads offer a way to get ready made 3D files to reappropriate into your own work.

MyMiniFactory, Scan The World <https://www.myminifactory.com/scantheworld/> is an example of a resource that offers incredible material. This website offers scanned 3D files of objects housed in museums from around the world, such as the V&A London, the Louvre Paris and the Metropolitan Museum, New York.

Coding

3D Drawing programs offer a visual interface but behind the screen a 3D file is made up of computer code. It is obviously quite possible to generate 3D files directly in code. Either a 3D mesh is generated that is then sliced in a slicing program to give .gcode. Gcode is the code language that 3D printers understand. This can be generated directly out of a computer coding platform.

Processing <https://processing.org/> is the coding platform that I use. Developed with visual artists in mind this open source platform makes use of the Java computer language and also has a large and supportive online community. Dan Shiffman's YouTube channel, Coding Train is an invaluable educational resource <https://www.youtube.com/user/shiffman/featured>.

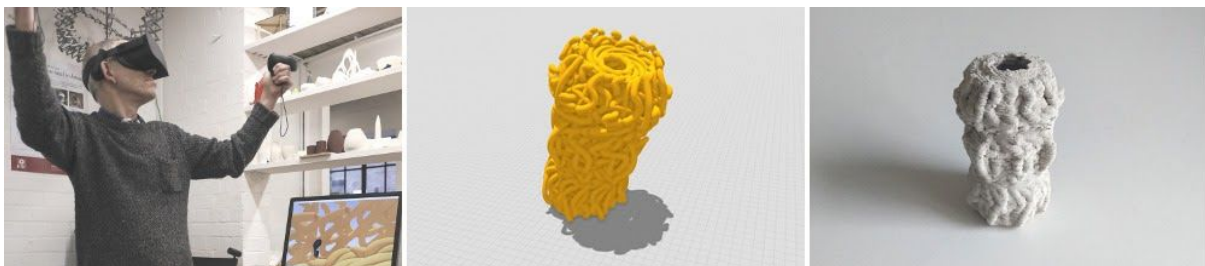
Augmented Reality

Virtual reality headsets such as the Oculus Rift offer 3D drawing capabilities. As this simulated 3D technology develops it will offer more and interesting whole body experience to working in virtual space. What I have done so far I prefer to consider as augmented reality, rather than virtual reality as I have used the technology to draw 3D objects that are realised as physical objects.

Vive VR <https://www.vive.com/uk/> system offers **Tilt Brush** <https://www.tiltbrush.com/> that at the time of writing is more of a painting program but will produce 3D files but is rather basic. It saves as a .fbx file type that can be opened/imported into the Blender software for editing and preparing for 3D printing.

Gravity Sketch <https://www.gravitysketch.com/> is a commercial software for VR 3D modeling and is an interesting software and 3D files can be exported as .obj file type.

Oculus Rift VR equipment has its own 3D modeling program called *Medium* that is a voxel based (kind of 3D pixel) software that makes easier files to edit afterwards than the Gravity Sketch or Tilt Brush type files. Currently *Medium* is the most interesting VR software in my opinion.



Hints and Tips on Preparing 3D Files

Thickness

The thickness of a print wall is defined by the thickness of the extrusion. For clay extrusion printing you should not give your model a wall thickness. If you do the printer will possibly try and print the inside and outside wall depending on the settings in your slicing software. When modeling for these machines you need just defining a single wall path. If you do want a double, triple or thicker wall this is set in the slicing software. The default will be for the path you draw to be the centre of the extrusion or combination/multiples of extrusions.

File size

It makes sense that the quality of detail in your 3D file is in keeping with the fidelity of your printer. The size of the nozzle in conjunction with the slice height will determine the detail of your final print. There is no point in having a high quality file, with a large memory size that could give your equipment problems to process, if the final clay output is going to be quite coarse. Blender offers a decimate Modifier that will reduce file size

Watertight file

As I have said a well drawn file produces a better print. Think about the *path* that the printhead will follow and the more continuous that this movement is the cleaner will be your clay print. It is preferable to have a so-called *watertight* 3D file. By this it is meant that there are no holes in your shape and that all the surfaces form one continuous shell. Slicing programs are improving all the time at dealing with problematic files but not all 3D files will slice perfectly.

Surface norms

Each computer drawn surface has a front and back referred to as *normals*. Normals can give problems when slicing files. All normals should be facing the same direction, either inside or outside. In the edit mode of Blender there is a setting to correct normals to one direction.

Added Support

Mentioned earlier, support for overhangs made in slicing programs is often not possible to print in clay. The answer is to add support shapes in your 3D model when drawing the model or alter the model to avoid overhangs if possible. For example if printing a portrait, under the chin often overhangs too much for the layers to build out. In your modelling program build in a narrow 45 degree wedge under the chin that will print and then once the clay gets stiff this can be cut away and cleaned up.

File Types

Like the *.jpg* has become the standard for 2D photographs, the *.stl* file type has become the standard for 3D models to be printed. More than likely you will be looking to *export* a *.stl* file from whatever program you have used to generate your 3D file. This program will have its

own *save as* file type and will probably not be transferable between programs. The **.obj** 3D file type is also quite popular and like the .stl file type can be read by most Slicing programs.

Slicing programs produce a **.gcode** file. Gcode is the computer language used by many computer guided machinery such as CNC machines and 3D printers. At its most basic it is not too complicated and a .gcode file can be opened in a text editor. It is not too difficult to read and understand. Like driving a car, you should not have to look under the bonnet to just work the machine but if you are interested in how things work .gcode is not complicated. Each line of code sends a command to the machine and the G1 command makes the printhead move from one xyz position in the graphic three dimensional space defined by your print area to the next xyz position. F represents the speed the machine will move at and E is the amount of material to be extruded by the printhead during each move.

Slicing

Once you have your 3D model it will need to be cut into layers or slices so that the 3D printer can build the shape. There are a number of Slicing programs available, both free to use and proprietary. Individuals will have their preferences but to get started **Cura** <https://ultimaker.com/software/ultimaker-cura> from Ultimaker will do you well. Slicing programs are predominantly developed for plastic printers so there are no preset or default clay settings. You will need to set up your own profiles that will work with clay.

Note: See section on setting up [Curra for clay printing](#).

Clay for 3D Printing



All clay can be used for 3D printing but some clays print better than others. What you are looking for to start out is a clay that is not too sticky, that has a fine texture and dries quickly. As you get to understand the process you may well change to clays of different properties to suit your specific needs such as clays that dry slowly if you do long prints. If you are using a printer with a screw printhead do not use anything that is too rough or you will wear the screw in the printhead.

Using clay for extrusion 3D printing does not differ greatly in process from traditional ways of working with clay. In the printer soft and malleable clay is given shaped by building up layers, similar to one of the oldest and most basic ceramic techniques, coil building. Computer guided coil building might almost be a better description than 3D printing. Then, as is the convention when working in clay the object is dried thoroughly and fired hard in a kiln. If a glaze covering is desired it is normal to apply the glaze after this first firing and the object undergoes a second glaze firing.

Clay Types

Pure **red clays** are often sticky and if you have a printer where the clay is fed through a long thin tube it does not flow very well. If you have a print that stops and starts a lot then sticky clay also becomes very messy. Fine red clays are on the whole slow drying. Sandy red clay or red clay blended with other clays move through the printer system better and dry quicker.

Cream clays with a good amount of 0.2mm grog/chamotte (15 - 30%) seem to work well. (Grog/chamotte is an already fired and crushed ceramic powder/grit) The grog makes the

clay less sticky and as with the sand in red clay the mix moves through the system better. It gives a bit more structure to the mix to help printed forms to remain standing up. Grogged clays also dry more evenly and quicker. Many suppliers indicate on the packet label how much grog there is in the clay. A fine grogged stoneware clay is a good starting point.

I have printed a lot in **porcelain** as I want the glaze quality you get from porcelain. Porcelain however has a very high shrinkage so it is not the most rewarding clay to print with. Porcelain is not very plastic or sticky so it extrudes quite well and it dries quickly.

Note: There is a very comprehensive and detailed document on testing six clays in the appendix - [2 Testing of Six Clays for Extrusion 3D Printing](#).

Additives

I do not get involved in clay additives such as deflocculating agents, drying agents like alcohol or stiffening materials, for example paper fibre. There may well be advantages and I would not suggest you do not try them but I manage well without them and work on the principle of keeping things simple. Basic clay is just such a lovely material I do not think you want to mess with it too much.

Too much deflocculation certainly is not good. Deflocculation, as used in ceramic slip casting, makes the clay become thixotropic. When agitated the clay becomes more liquid and it will not work in a screw printhead. Even if using a direct flow extrusion the clay does not layer up well. Deflocculation can happen unexpectedly through using soft (alkaline) water, especially with porcelain clay. The solution is to mix a small amount of vinegar into the clay (5 ml per 10kg clay). There is a logic to using the clay prepared for slip casting for 3D printing as this clay has less water in it but then you get this thixotropic problem. Avoid slip casting clays.

In Italy working with WASP we did tests using bioethanol mixed 50/50 with the water used to soften clay out of a bag. I think there could be advantages as to the speed the clay dries but back in the UK, working in an enclosed space the fumes would be too much so I have not followed up on this. I prefer to use photographic lights or heat lamps or warm air blowers to help dry work while printing.

I have done some experiments with [additives](#) and the results can be found in the [appendix](#).

Clay Consistency

The clay for extrusion 3D printing needs to be soft enough to be fed through the machine but you do not want it to be so soft that it will not support itself as it builds up. There is no simple measure for what this consistency should be but something that looks like toothpaste is a good guide. The clay surface will look just moist but not saturated wet.

Different clays have different characteristics and absorb different amounts of water to become soft so it is difficult to offer exact numbers. It comes down to getting a feel for it and I still get it wrong. Excuse the earthy analogy but if you slap down handfuls of your prepared clay you want it to look like a healthy cowpat. If the pile slumps down like cowpats from cows that have been grazing on fresh spring grass it is too soft. If as you slap down handfuls of clay and there is no 'give' between layers then the clay is probably too hard. As I said, you want a good healthy cowpat quality. See [JK Drop Spike](#) for measuring clay consistency.

Hard or Soft Clay

Pressure, either using compressed air or a mechanical ram is used to force the clay through the system. Under pressure water gets squeezed out of a clay mix and the higher the pressure the more this happens. So while there is logic to using hard clay so that a print does not collapse, this will require more pressure to get it through the system and you will get more dewatering and the clay in your bulk container becomes even harder. Harder clay can also result in the printed layers not sticking to each other and delayering or cracking occurring. I tend towards soft clay that requires 4 to 5 bar of pressure to get it through the system. There is some testing I have done with regards to this subject in the [appendix here](#).

Mix from Wet or Dry

Clay can be prepared from bags of wet clay ready for use on the pottery wheel or from dry powdered clay. Clay particles are described as platelets and the water acts as a lubricant between the plates. It is this structure that gives clay its plasticity and different kinds of clays have different platelet sizes. Very fine clays like red clays are more sticky and do not work as well in extrusion printing as do larger platelet clay, like china clay.

While you can soften clay quickly with water it takes up to a day or more for the water to penetrate the finest particles. So clay that is mixed today will be that little bit harder tomorrow, even kept in plastic not because the water has evaporated but because it has been further absorbed into the mix. So softening ready wetted clay is preferable for immediate use than mixing from powder. Freshly mixed clay from dry powder often has a slightly granular texture and lacks plasticity and plasticity gives better print results. If you do mix from dry allow the clay to age as long as you can before use, days preferably weeks, even months.

Coloring clay

There is no problem printing with [stained](#) or coloured clays. Light coloured clays, so the colour shows through, are best to add ceramic colours too and choose a clay that prints well in the first place. It is important to get the colour thoroughly mixed so you will probably do well to soften the clay to the point you can sieve the mix and then allow it to dry again. Different naturally coloured clays can be blended to give more earthy colours.

If you layer clay in a clay container for printing you will find that once it extrudes there is not a clear distinction from one layer to the next, there is a blend. This is because the material moves faster in the middle of the extrusion and drags on the outer surface. For clear two or more colour printing you will need a clay container and printhead for each colour. [See](#) project by Tom Lauerman.

Clay Shrinkage

Different clay shrink different amounts. As clay dries and the water evaporates the clay shrinks but then it shrinks further in the kiln when it gets fired. The higher the temperature the more the clay shrinks. Normally clay shrinks equally in all directions but with 3D printed shapes the shrinkage is greater in the vertical than in the horizontal. I guess this has something to do with the layers being compressed together. As a guide stoneware clay shrinks about 13% in the vertical and 8% horizontally. Porcelain is up to 20% or more in the vertical and 16% horizontally. This is when fired to 1220 degrees centigrade.

There are results of shrinkage tests in the [appendix here](#).

Softening a bag of throwing/modeling clay by hand

Take a potter's wire cutter and cut the block of clay into thin (1 cm) slices. With a kitchen fork lacerate as much of the surface of each slice as you can and then dip the slices into a bucket of water and then pile them up. What you are doing is exposing as much surface as you can to water. If you have planned ahead and can leave the pile of slices overnight (covered in plastic) the clay should soften quite well. If you want to get on immediately it is time to get your hands dirty and squish handfuls of clay together and keep mixing by hand. You will probably need to spread the clay and lacerate it again with the fork and add more water.

I work on a non porous board, an old kitchen cupboard shelf is good and use a sponge to squeeze on more water. This hand mixing continues until you have the correct consistency (like toothpaste) and the whole pile of clay is the same. Mixing by hand enables you to feel for any lumps. For good prints the clay paste should be perfectly smooth. Any inconsistency of texture will show as thick/thin extrusions on your print. When mixing the clay you want to avoid getting air bubbles trapped in the paste. Scraping layers of paste with a cupped hand and slapping it into a pile seems to work quite well.

There is a basic video [here](#) of preparing a batch of clay by hand.

Mechanic clay preparation

Anybody with a deairing clay mixer pugmill is in luck as these expensive pieces of clay studio equipment do a good job. These machines will mix from powder or you can add cut up

throwing/modeling clay. I have seen bags of cut up throwing/modeling clay mixed (with extra water) to a stiff paste consistency in a pasta mixing machine. You need a mixing machine with a lot of strength. In my view mechanical mixing is only worth doing for large batches as the time it takes to clean the machinery afterwards outweighs the time it would have taken to just mix by hand.

For mixing from powder in bulk I use a builders double shafted mixer, as used for plaster but it will only mix to a consistency much softer than I want for printing. I leave this mix uncovered to slowly dry out until the required consistency. This also gives the mix time to thoroughly wet through and age. This is the technique I use for large batches of clay. The paste is stored in plastic bags and I will hand mix each batch before use.

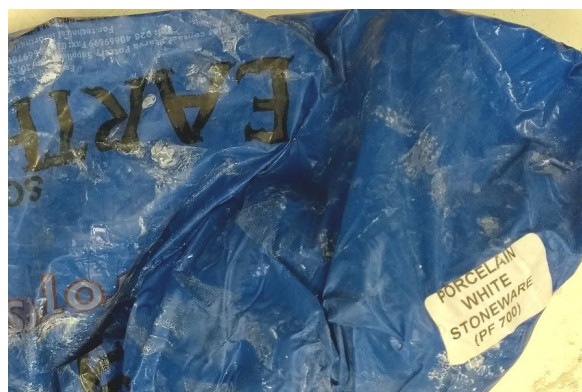
Clays known to 3D print well

Porcelain

In the UK I have printed successfully with both the Royal and Special porcelain bodies from Valentine Clays. I have tried Valentine's Parian Body, that has a reputation for translucency but found the unglazed fired results look rather like printed white plastic. More seriously however was that forms cracked a while after coming out of the kiln that is normally the sign of tensions in the clay body. I have abandoned using Parian.

In France I have used Limoges porcelain with equal success to the UK porcelains. Limoges porcelain is a case where I will add a bit of vinegar to the mix to stop it from becoming thixotropic.

Stoneware Clay Recommendations



(UK) Scarva Earthstone Professional White Porcelain Stoneware PF700 - available from a number of suppliers.



(France) Ceradel White Stoneware CG811 - this has a high proportion of 0-0.2 mm grog/chamotte in it.



(Spain) Sio-2 Artemis White Stoneware 164-ARTE - available in the UK from Potclays.



(Sweden/Denmark) Red 1122 - available from cerama.dk and cebex.se

Note: Charlie Stern has used this clay very successfully.

3D Printing Clay recipe

If you look at the details on the bags of clay illustrated above you can begin to see what will go into making up a good clay printing receipt. You are looking for a range of particle size in the form of clay and grog.

JK Print Mix

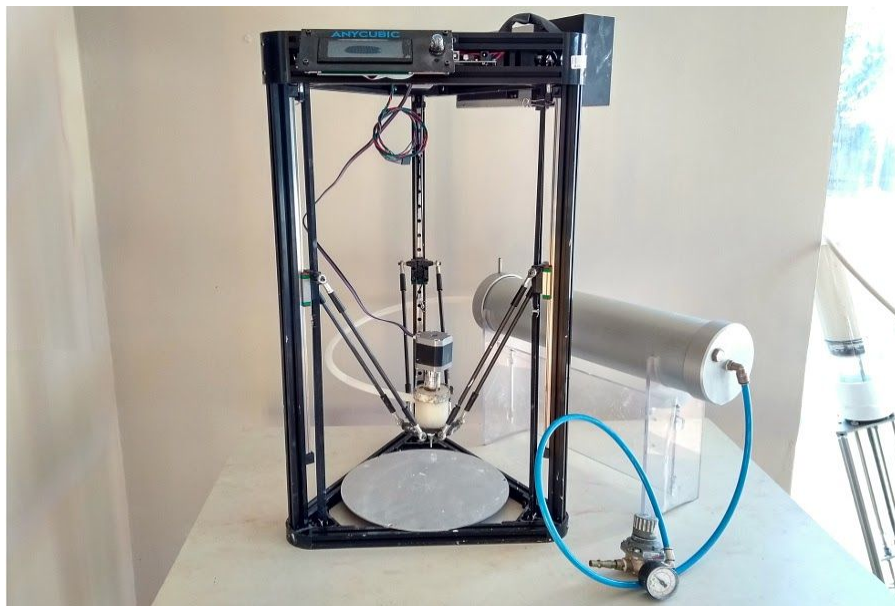
Hyplas 71 Ball Clay	35
Grolleg China Clay	20

FFF Feldspar	15
Fine Molochite (0 - 0.2 mm)	30
Bentonite	2

Note: For printing larger works half the Molochite will be medium grade (0.5 mm).

I have written further on formulating a 3D printing clay body in the [appendix here](#).

Convert an AnyCubic Kossel for Clay 3D Printing



In 2013 I published the design of the Make Your Own JK Delta Clay printer. At the time there were not the cheap table top 3D printers that are available like now. Now it is probably cheaper to buy a kit than the cost of just the parts to make a JK Delta Self Build. Clay 3D printing technology has also moved on since 2013 so this guide explains how to put together for yourself a reasonably priced clay 3D printer by converting a plastic extrusion printer.

In 2017 Dries Verbruggen documented converting an AnyCubic Delta printer to operate with a Glue Gun type printhead as used on the JK Delta Self Build. see <https://wikifactory.com/+Ceramic3DPrinting/forum/thread/MzE2OTg?categoryId=Mg>. In this documentation I cover the conversion of a similar AnyCubic Kossel Delta printer to work with a bulk clay container and a stop start screw controlled printhead. The passing of time has caught up with us and AnyCubic no longer makes this printer but a bigger delta 3D plastic Printer called the Predator. With a build volume of 37 cm diameter 45 cm height this is a better sized machine than the old Kossel. While I have not had the personal experience I have heard of others converting the Predator as in my documentation below. I hope this information would also be relevant and useful for converting any other plastic printer to print with clay.

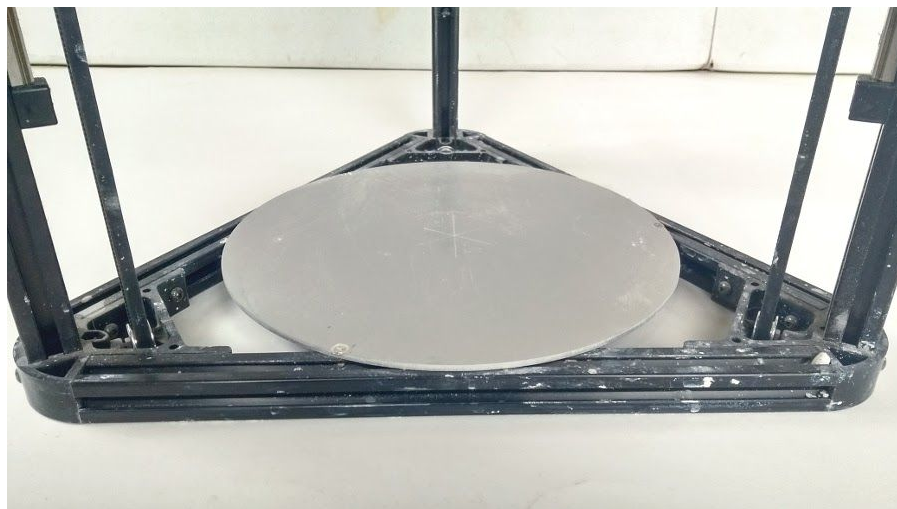
As you are probably aware I like to break up the thinking of clay 3D printers into rig, printhead and clay delivery. The rig is the computer controlled machine that will move the printhead. The printhead is the mechanism that deposits the clay and is relatively simple to make and instal. Then the clay delivery is the process of getting the bulk clay material to the printhead.

Rig - The AnyCubic Kossel

When you get your AnyCubic Kossel make sure you get the version with linear bearings that run up and down the vertical frame and not the version with wheels. I think the wheel version is no longer available, but the wheels were not very stable. The Build area is described as having a 23 cm diameter and 30 cm height.

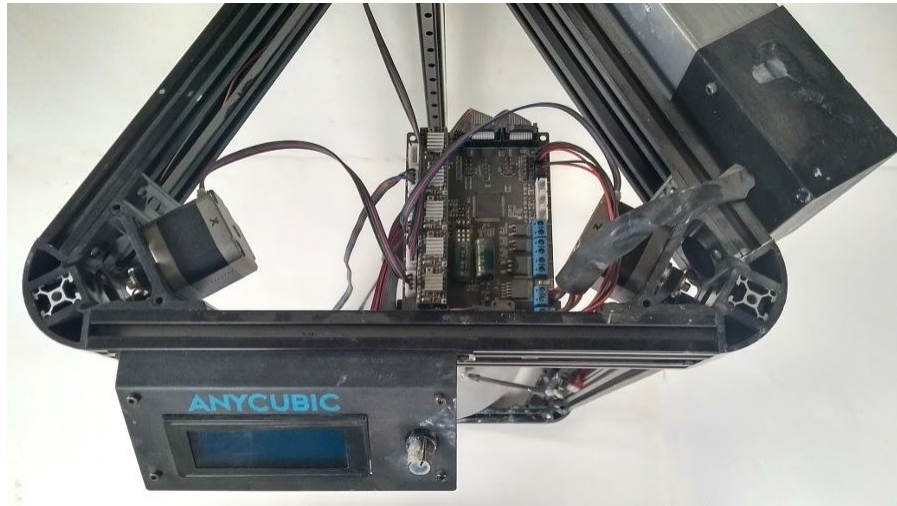
Your printer will arrive as a flat pack partially assembled kit. I will not go into detail of the build as the user manual is very good. What I have done is to offer photographic documentation of my build as I have turned the rig upside down. In the conventional design all the electrics and controls are at the bottom. When working in wet, then dusty clay this is not a good idea, hence I have put all the electrics at the top.

Moving what was the top frame to the bottom is straightforward. The base plate had a heater attached underneath that I ripped off. You will find there are base plate brackets on the bottom frame (that will go at the top) and I just removed these.

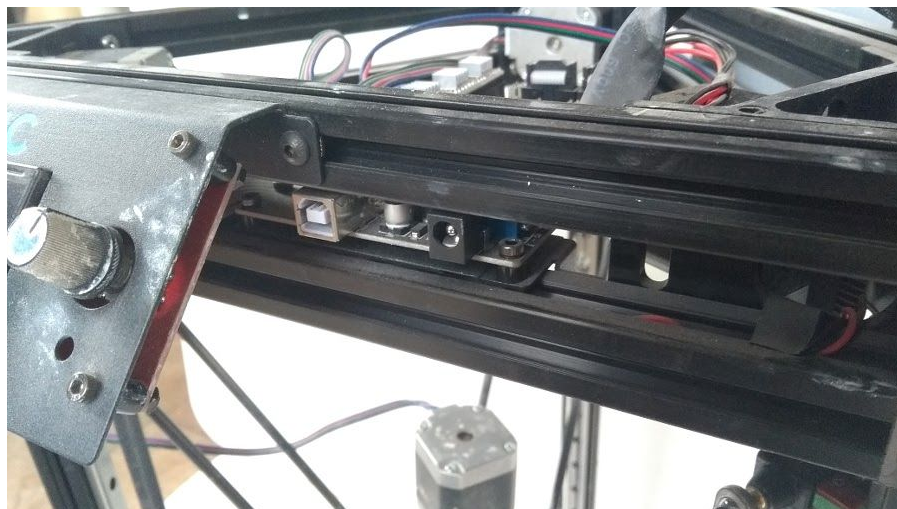


One thing to take note of when you put the pre-assembled bottom frame at the top is that the distance the vertical beams slide into the top and bottom frame. When attaching the Linear slider bars to the vertical aluminum frame bars the positioning gap must now be greater at the top. Space must be allowed also for The Top End Stop bracket. This makes sense when you have the parts. Once you have all the electrics at the top the wiring from the end stops does not have to be pushed through the aluminium beams. The bottom frame

that is moved to the top is not flipped, it is just moved up. As you will see I have the display screen on the same side as the electronics mainboard so the wires can be kept neat.



I did move the mainboard along so I could access the usb input next to the display screen. The display screen is mounted left of centre.



The Power supply I mounted horizontally and up away from any water. The wires just reach the mainboard.



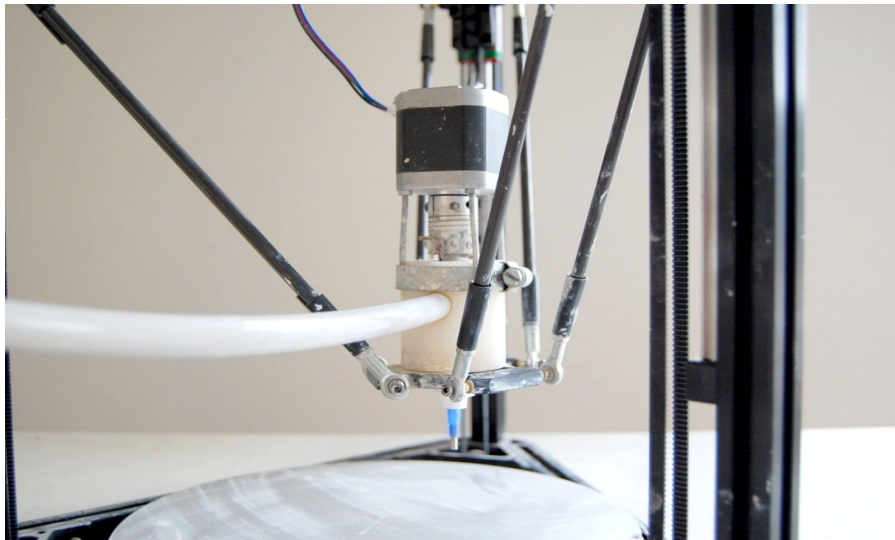
Follow the build instructions until you get to the Filament holder and Extruder motor. The filament holder is not attached and the Extruder motor, minus the mount is used to make the Clay Printhead - see next section. The Anycubic Plastic printhead is removed from the printhead mount plate and your Clay Printhead will later get attached to this plate. The plastic printhead is not used and is pre-assembled on the printhead plate that is attached to the diagonal rods.

The wiring diagram is very good in the user manual. Obviously the Hotbed, Heated Nozzle (for plastic) and the Fans are unused.

I have set the z level in the Arduino Firmware on my machine so the printer lowers to the same point each time. Anycubic offers a z Leveling sensor and has thorough documentation on how to set the z level. This can be done once you have sorted out your clay printhead and know what length your nozzle will be. I keep the z to one height and change the thickness of boards underneath if there is a change.

Clay Printhead

I have document how to make your own printhead in another [section below](#) and here - <https://wikifactory.com/@jonathankeep/self-build-clay-printhead>



You could also look at -

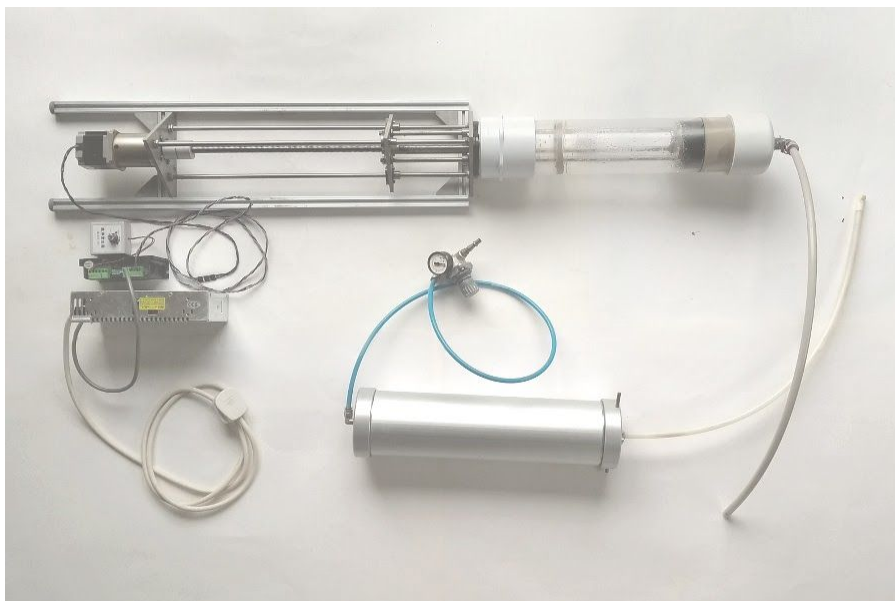
Stoneflower Clay Printhead Kits - <https://www.stoneflower3d.com/>

Stoneflower Open Source page - <https://www.stoneflower3d.com/opensource/>

WASP Clay Printhead Kits - <https://www.personalfab.it/en/shop/clay-extruder-kit-2-0/>

Clay Delivery

In the introductory image the clay delivery is the large aluminium containers (right back) and this will be attached to an air compressor. The compressed air obviously pushes the clay to the printhead.



I do also have a mechanical ram per kind favour of [ClayBot](#) so I have run my Anycubic with both a ram clay delivery and the compressed air delivery. Both work well but I prefer the simplicity of the air system. I would use the same consistency of clay in either system. In the photograph above the air pressure container (lower middle) holds more clay than the ram above it and is easier to clean and maintain.

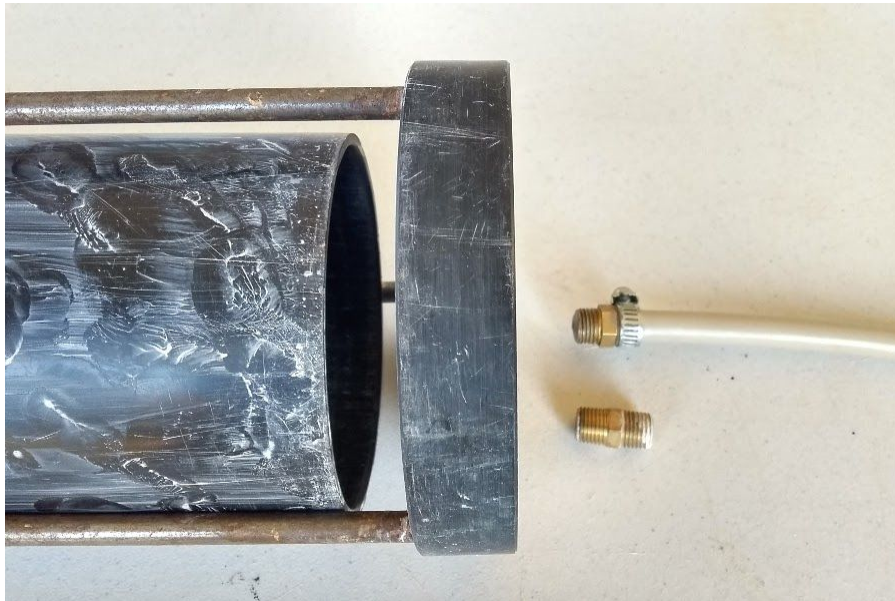
With four bracing rods placed around the circumference of the endplates this would bolted up more evenly.



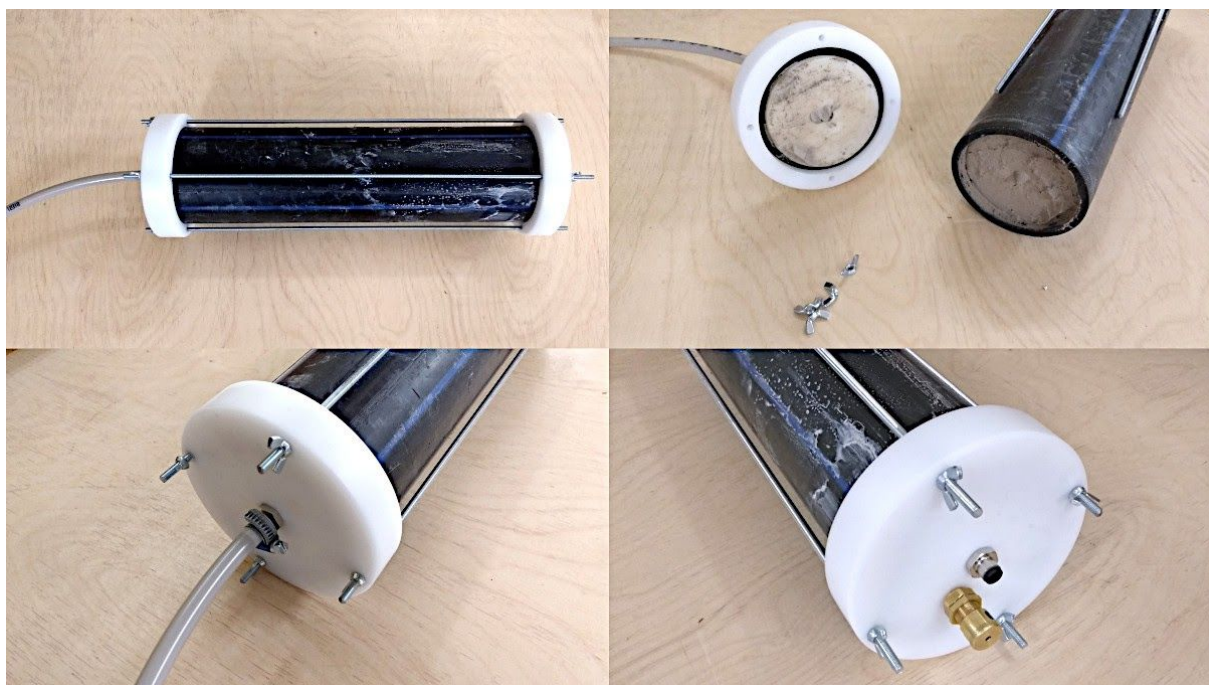
There are two end caps made of engineering nylon. I do not know what the main black container is made of but it is said to be good to 8 Bar of pressure. Then there is a plunger (white in photo), also made of engineering nylon that moves up and down inside the black pipe. Same in the aluminum container. This pushes the clay and seals with two O rings so that the air pressure does not push past the plunger. In each end plate there is a round slot that the black body pipe fits down into. In the slots are more rubber O rings that seal the air at one end and the clay the other. These end plates are 25 mm thick and the slots 18 mm deep.



This is the endplate where the air comes in. A push connector holds the air line coming from the pressure regulator. I print at between 4 – 6 bar of pressure. The brass fitment to the right on the end cap is a safety pressure valve that I am told will release pressure above 8 bar of pressure. A very important safety feature.



At the clay end there is a nylon pipe that carries the clay out of the container and to the printhead. I use Nylon Pneumatic Air Line Tubing, clear/natural with an outside dimension of 12 mm and inside dimension of 10 mm. This also comes with a 9 mm inside diameter and this pipe is then too stiff. The 10 mm inside is more flexible, that is good but also kinks occasionally and needs replacing. The length of this feed pipe is 70 cm long. It is important that the flow of clay is not restricted in any way. Keep all inside diameters to 10 mm. You can see the brass double ended threaded connector that I use in the photo, right. This pipe attachment is specific to the needs of the clay printer and is not part of the original supplied boiler gun.



A Clay container made in Reykjavik to the above description. A very thick heavy duty body pipe was sourced. Obviously this could be substituted to a metal pipe, preferable aluminium for weight. Having seen this

container working well my only comment would have been that the four long bracing, tightening, threaded rods could have been a bit heavier.

Some links to Boilie guns -

G S Boilie Guns - £120.00

<https://www.gumtree.com/p/fishing-tackle/g-s-boilie-guns-the-6kg-nighthawk-boilie-gun-new-/1341591454>

Boilies Making Gun, Air Bait Boilies Gun KG.6 - £138.00

<https://www.ebay.co.uk/itm/BOILIES-MAKING-GUN-AIR-BAIT-BOILIES-GUN-KG-6-3-nozzle-SAUSAGE-GUN/122822395372>

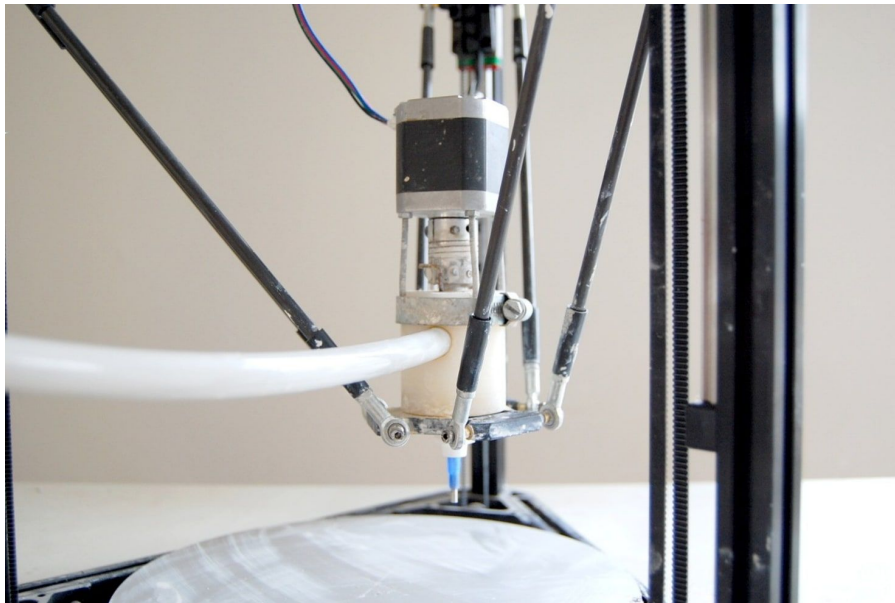
Parts List for (old) Make Your Own Clay 3D Printer

http://www.keep-art.co.uk/Journal/Delta_3DPrinter_Parts_Nov15.pdf

Note: While these containers have cone shaped extrusion nozzles it is best if your clay pipe leads straight out of the end plate. In the past I have attached the clay pipe to the cone but clay becomes compacted on the inside of the cone. It is best to have a flat plate on the container end plate and attach the 12 mm nylon pipe directly into this plate.

Self Build Clay Printhead

A guide to making an auger, screw type stop start or retraction printhead for clay 3D printing.



The overriding design principal with this clay printhead was that it would be easy to take apart, to clean and maintain. I do not like plastic printed parts as I have found them not to be strong enough and why go to that length if the parts can be kept this simple. My judgement for the design decisions is not very measured but a mix of intuition and suck it and see, combined with what parts I can source easily. I'm not claiming this is the ultimate design but it works for me and is disarmingly simple. This printhead can be used with a mechanical ram type of clay delivery or with a compressed air clay delivery and can be fitted to either Cartesian or Delta type 3D printers.

The Printhead

The body of the printhead is made of engineering nylon. My choice of this material is because clay is incredibly abrasive so I wanted to avoid metal parts (except for the screw) and with my basic hand tools I can easily cut and drill this material. Further I use different sized screws depending on what scale of printer and size of extrusion or nozzle I intend to use and so the body can be made to fit the screw. The solid nylon bar can be cut with a wood saw and then drilled vertically for the screw size and from the side for the clay delivery inlet pipe. This cylinder of nylon also mounts comfortably onto most 3D printers using M3 bolts that will self thread into appropriately sized drilled holes.

Parts

- 1 x Nema 17 Stepper Motor
- 4 x 70mm M3 threaded rod

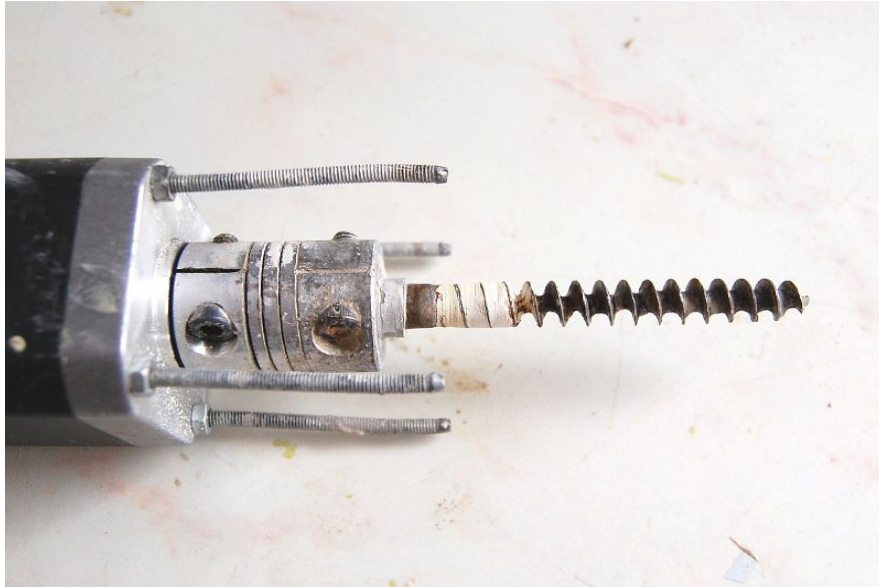
- 4 x M3 nuts
- 1 x Flexible beam coupling
- 1 x 6mm roofing screw
- 1 x 55mm length 45mm diameter engineering nylon rod
- 1 x Worm drive hose clamp 50mm dia
- 1 x 25cm natural nylon pneumatic airline pipe 12mm OD/10mm ID
- 1 x ¼" male Luerlock connector fitment
- 1 x Luerlock dispensing tip – Olive Green cut to 2.5 mm opening

The stepper motor I use is a SY42STH47-1684A High Torque Hybrid Stepper Motor from <https://www.zappautomation.co.uk/electrical-products/stepper-systems/stepper-motors/nema-17-stepper-motors/sy42sth47-1684a-high-torque-hybrid-stepper-motors.html> . Four lengths of M3 threaded rod are bolted into the stepper motor to connect the motor to the nylon body. The length of these are determined by the beam coupling that is used to connect the motor drive shaft to the auger screw. (Different sized beam couplings are available). A hose clamp is used to clamp the M3 spacer rods to the nylon body cylinder. The motor can then easily and quickly be removed from the main printhead body for cleaning. The motor wires plug into the E pins on the Ramps board with corresponding driver and power adjustment screw. This motor is the equivalent of the feed rate in plastic printing and is controlled by the .gcode generated in your slicing software.

The nylon body cylinder is the same diameter as the glue gun extruder of the original JK Delta design so that it could be interchangeable. If this is not necessary a body diameter of around 40 – 45 mm is a good idea so there is enough body material to drill the holes necessary for the auger screw and clay inlet pipe. I have varied the nylon cylinder length depending on the weight of the auger - between 50mm length for a small 6mm screw and 70 mm long for a larger 8mm screw. The screw length is determined by the length of the body cylinder. I aim to have the tip of the screw ending as close to the nozzle as possible. Obviously a hole the diameter of your screw is drilled in the vertical centre of the nylon body cylinder. A wood/metal drill bit will do this. I believe in the auger screw being a snug fit in the printhead.



Although the turn of the screw will be driving clay down I feel it is favourable to have the clay feed/delivery under more pressure than the screw will naturally take to help force the clay extrusion. To stop clay rising up the screw shaft I seal the upper thread with hardening putty that forms a tight fit. Clay sometimes oozes up but it is easily cleaned away. If there is excessive ooze then the clay feed/delivery is too great. This is how I judge the air pressure or ram drive rate, whichever I am using.



Feed pipe

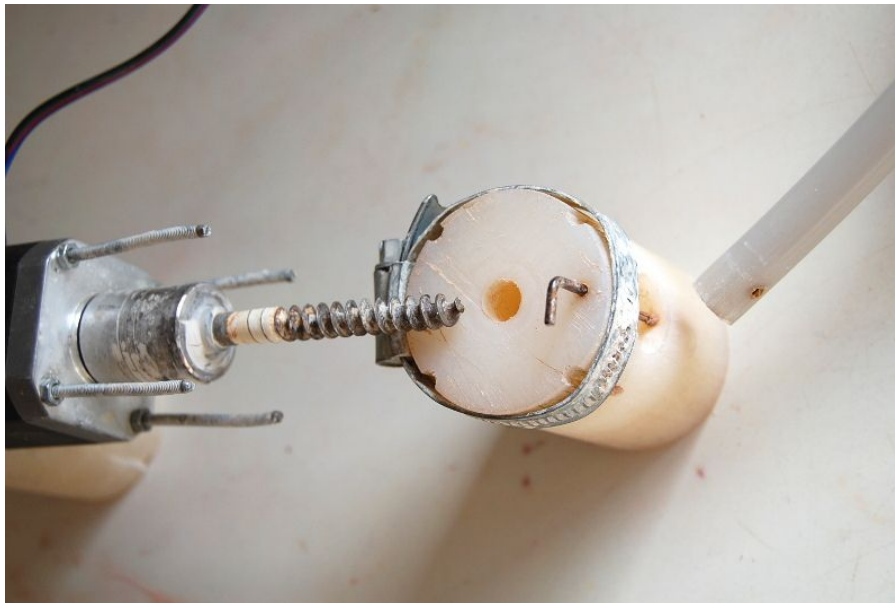
I have tried various feed pipe attachments. Push connectors clog up with clay and stop working. Threaded connectors are a pain to undo. This simple wire pin through the pipe does the job. The clay feed pipe I use is 12mm natural nylon pneumatic airline pipe available from

<https://www.ebay.co.uk/itm/METRIC-NATURAL-NYLON-PNEUMATIC-TUBING-AIRLINE-PIPE-P-MT-/250806057958?var=&hash=item80104bfe99>. I use this as I believe it to be more

slippery than softer types of plastic pipe. Watch that you get 10mm inside measurement or the pipe is too stiff and you need as big a diameter as possible. A downside is this pipe does tend to kink. (Outside Dimension 12mm, Inside Dimension 10mm)

A 12mm hole is drilled into the side of the nylon cylinder body to take the feed/delivery pipe. This hole is drilled at least the width of your hose clamp down from the top of the nylon body and far enough in to connect with the vertical screw channel.

With the feed pipe in place I drill a vertical hole to take the wire pin that keeps the feed pipe in place. This vertical hole I drill the full length of the body cylinder so clay can be cleaned out of this hole. If not the pin will not go down after a while. An old bicycle spoke makes for a good pin.



Nozzles

I use a range of nozzle sizes depending on what I am doing so I have kept to a standard $\frac{1}{4}$ inch threaded outlet hole underneath the printhead. The central hole is drilled bigger and a $\frac{1}{4}$ " tap used to create the thread. This gives the option of attaching a male Luerlock fitment <http://www.adhesivedispensers.co.uk/TSD931-7C.htm> that will take the range of Luer Lock dispensing tips <http://www.adhesivedispensers.co.uk/tips.htm> or larger $\frac{1}{4}$ " dispensing nozzles <http://www.adhesivedispensers.co.uk/nozzles.htm>. I do like a metal nozzle tip so I customise my own nozzles. Nozzles will range from 1.2mm up to 4mm with 2 mm being my workshop standard.



Screws

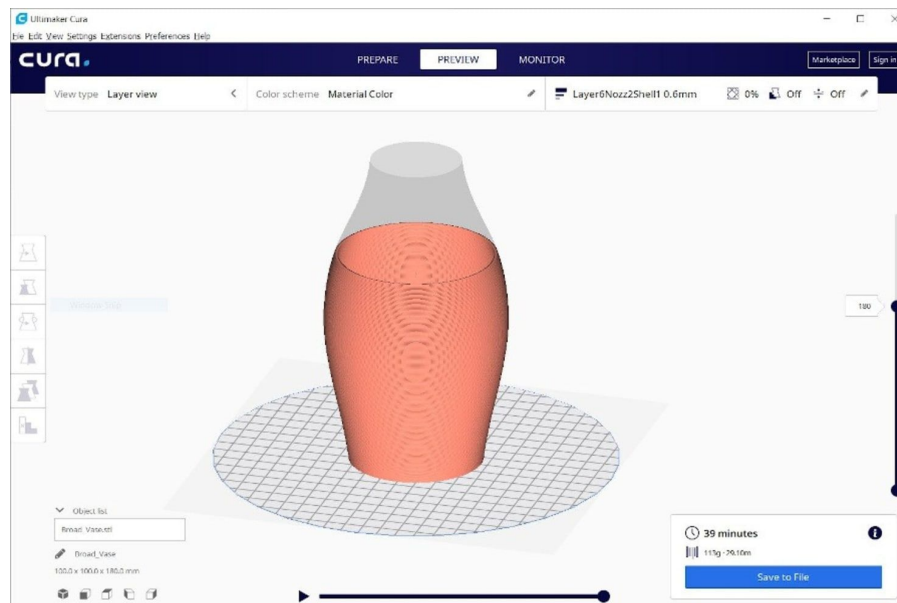
The screws are standard and I get them from my local hardware shop. To improve the efficiency I grind the tread deeper with an angle grinder and then cut the screws to the

required length. I am undecided whether a busy or lazy thread is preferable. My current thinking is that for a small nozzle (1.6mm olive colour) a busy thread is good to grip and force the clay through the small aperture backed up by the pressure of the clay coming from the clay feed/delivery. For big nozzles (3 -4 mm) the restriction is less and the pressure from the clay feed/delivery almost pushes through the clay so a lazy screw is preferable to help the clay on its way.



Cura for Clay 3D Printing

Setting up Cura version 4.3 for Clay 3D Printing - this is what works for me, November 2019.



Download from <https://ultimaker.com/software/ultimaker-cura>

Introduction

On setting up you will probably be asked to Add a printer. Use Add non-networked printer, then I would suggest just go with the first Ultimaker printer that is selected and go add. By doing this you can begin to see the layout of the software and find your way around. There are loads of YouTube videos on how to use Cura, so look at those but here I want to go through the setting up of Cura for clay 3D printing. This is only a basic setup and as you become familiar with the program you will find that it has much more to offer.

Layout

Top left is a horizontal menu that I do not use much except for Settings. Centre top is Prepare and Preview that I do use. Monitor is for networked printers that I do not use. Below this, left is a folder ikon and I use this to load 3D files for slicing.

Next is the Printer Name that is active and a drop down arrow offering the names of all other printer profiles as they get set up.

Next, going across horizontally is the Material that is selected and similarly a drop down arrow that will show all other materials with a profile compatible with the selected Printer settings.

Finally moving horizontally across are the Print Settings. Clicking on this opens Profile options and settings. A second click closes the drop down window.

In the main part of the layout is a visualisation of the selected Printer and once you open a file by clicking on the Folder Icon (top right) and selecting your .stl or .obj file (other file types are available) you will get a visualisation of your 3D object.

Setting up Printer Profile

A new printer can be added from the Settings menu (top left) or clicking on the drop down arrow in the selected Printer box.

Select Add a non-networked printer.

Then go down and select 'Custom'.

I will be describing setting up a Delta Type printer such as the JK Self Build, WASP Delta or an AnyCubic Delta, so select 'DeltaBot' in the drop down from the arrow next to Custom, and Add.

If a 'Machine Settings' window does not appear go to 'Settings' (top right) and in the drop down at the bottom is 'Configure settings', click. This opens the main 'Preferences' window that you will get to know well. In the left hand column go down to Printers and select DeltaBot if it is not already selected. Note there is an 'Activate' tab below 'Printers' heading, should you change printer selection.

There is a 'Rename' tab so you can give a custom name to your printer.

Now set up the properties of your newly named custom machine by clicking on 'Machine Settings' (right). A 'Machine Settings' window opens.

Under Printer Settings in this new window put in your printer dimensions in mm -

JK self build: x width 180 mm, y depth 180, z height 300.

WASP 2040: x width 200 mm, y depth 200, z height 400.

AnyCubic: x width 180 mm, y depth 180, z height 250.

All the above printers 'Build plate shape' are 'Elliptic'.

All have their 'Origin at Center' so click the box if not already selected.

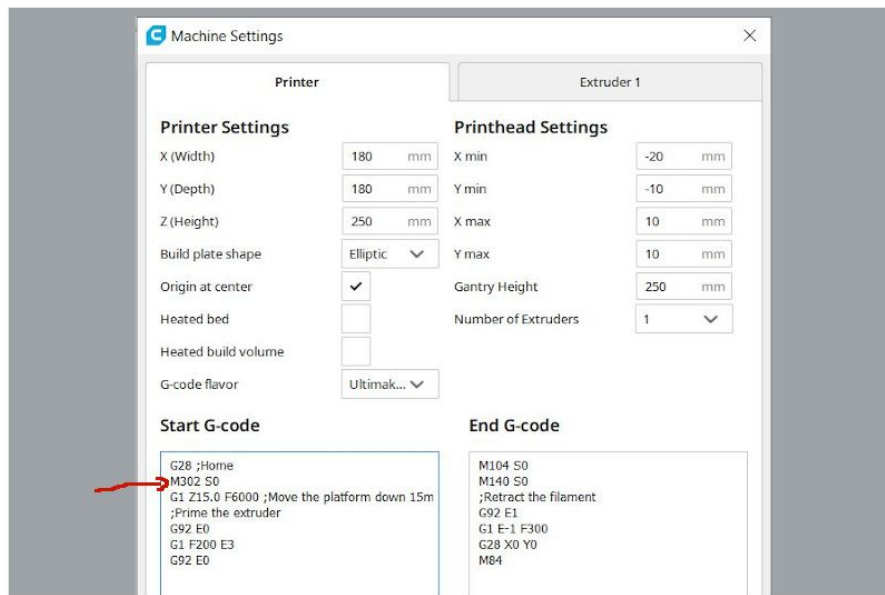
None of the printers have Heated bed so do not click box or deselect.

None have a Heated build volume so do not click the box.

Note: For WASP machines set the G-code flavour to Marlin.

Note: For JK Self Build and AnyCubic Delta Clay conversions, set the 'G-code flavour' to 'Ultimaker 2'. (The other flavours put in a M105 temperature checking gcode that will not let a cold clay print start!)

Further, for JK Self Builds and AnyCubic conversions, stretch out the 'Machine Settings' window so you can clearly see the 'Start G-code' in the lower left box. Under G28 ;Home add a new line of code that reads; M302 S0. This setting allows a cold extrusion and by adding it to the Start G-code it should be included each time you slice a file for printing. (There is no need to make this customisation for WASP printers)



Printhead Settings

I leave all these settings as default for whatever Printer I am setting up (basically I do not know what this is all about and it has not caught me out yet)

Extruder Settings

Note: If you are setting up a printer where the extruder is not controlled by code as in the JK Self build, that uses glue gun type cartridges and the flow is switched on and off manually, you do not need to worry about the Extruder settings.

Note: If you have a printer with a screw/auger printhead you must set up the Extruder 1 by clicking on the tab to the right of the Printer tab.

Enter a Nozzle size. A 2 mm nozzle is a good starting point. The nozzle size regulates the flow of material as controlled by the screw/auger/feed speed. So if you will print with different nozzle sizes you might set up different Printer profiles. However I think this nozzle size setting can be overwritten in the Print settings later on.

Compatible material diameter - this is where things can become a little confusing in this version of Cura. I like to use a material diameter of 1 mm. It is a simple round figure to balance any other settings against. However when you come to set your material for this Printer profile no materials will be visible as there are no default materials that have a diameter of 1 mm in the Cura library. The answer is to make a custom material with a diameter of 1 mm. When you have the Ultimaker Printer selected that you will have implemented in the introduction, all the default materials will be visible. I will go through this when we get to the materials settings.

The material diameter is important because if you set a large number the software will think there is a lot of material being fed and the screw/auger flow will be slow. The smaller the material diameter the faster the software will set the feed rate to pump through enough material. So later in your Print settings if you find the default 100 % flow rate in the material

settings are way off this can be balanced by altering the material diameter. For a WASP 2040 the material diameter is 1 mm but for a WASP 40100 I have it set at 2 mm. Remember we are customising a software that is designed for plastic printing to work for clay. Close the Machine Settings

Setting up Material Profile

(If you have a JK Self Build or a Printer where the printhead extrusion and material is not controlled by gcode you do not have to do this. The material settings like the Extruder setting are all irrelevant)

For WASP printers and any printer with a screw/auger printhead you will need to make a Natural Clay material profile.

Select the Ultimaker printer I suggested you implement in the introduction.

Go to 'Settings' (top left) and in the drop down click on 'Configure settings visibility'.

Select 'Materials' in the left column.

Click on the Create tab under materials and a 'Custom' category is added with a generic, probably PLA material. Now edit this material.

So under Information (right in the pop up window) give a 'Display Name' such as Natural. Under 'Brand' call it Clay and Material Type also call it Clay. Click on the Colour box and select a good clay colour of your choice.

Further down under Properties give a Diameter of 1 mm and press enter. You get a warning saying the filament diameter is not compatible with the current extruder and do you wish to continue. Yes you do because this material is for your Delta Printer that does have a 1 mm filament.

The material now disappears because you have the Ultimaker Printer selected. Close the settings preference window. Go to the Printer selection drop down arrow (upper left). Select the Delta Printer you set up earlier. Move over to the Material drop down arrow and there is a further drop down arrow, and if you are lucky there is your Clay, Clay, Natural as you move the cursor out.

However there is one more thing that is needed in the Clay Profile and that is to set all the heating temperatures to 0. Go, Settings/Configure setting/Material (top left) and in the Preferences window that opens, making sure your new clay material is selected find the 'Print setting' tab to the right and set all the 'temperatures' to 0. The 'Retraction Distance' and 'Retraction Speed' can be left as default. Set the 'Fan Speed' to 0 as there is not a cooling fan.

Setting up Print Profile

Click on the Profiles banner to the right of the Material banner. This opens the drop down window of 'Print settings'.

In the drop down menu to the right of 'Profile' select the 'Extra Coarse - 0.6mm' profile to edit to your preference. The print settings profile is broken into a number of subheadings. Subheadings can be opened and closed with a click on the subheading name banner. Hover over any setting in a subheading and a small but useful description window appears. Hover over a subheading banner and a setting cog icon appears to the right of the banner. Click on

this cog icon and the main 'Preferences' window opens. Here you can select which Settings can be made visible within any subheading. The Profile presets of which we have selected Extra Coarse help refine this mass of possible settings.

Quality

Layer Height - with a 2 mm nozzle I use between a 0.6 - 0.8 layer or slice height. (0.8 mm)

Initial Layer Height - half normal layer height helps the first layer stick down. (0.4)

Line Width - will be set by the nozzle width set in the Printer profile. (2 mm)

To simplify your Profile the WallLine, Outer Wall, Inner Wall, Top/Bottom Line Infill Line Width and Initial Layer Line Width could all be deselected in the Preferences/Setting Visibility by clicking on the Quality cod icon and then deselecting the items check box.

Shell

Wall Thickness - this should be the same as your Line Width that is the same as your Nozzle width. (2 mm) If you want your object to have a double, or triple thickness make this setting twice or three times the Line Width.

Wall Line Count - by default this will be set as 1 if the Wall Thickness is set the same as the Line Width. Change this Wall count to 2 or 3 and the Wall Thickness becomes greyed out.

Top/Bottom Thickness gets taken from the Layer/Slice Height. I tend not to print a base or a top so I set this to 0.

Top Thickness - without infill you are unlikely to print a Top in clay. I set it to 0. Once set to 0 I hid the setting.

Bottom Thickness - I tend to print onto a slab of rolled out clay so do not print a base. If I do print a Bottom/Base, I will print at least 3 layers so your base would be Layer Height of 0.8 mm x 3 = 2.4 mm.

Optimize Wall Printing, Fill gaps, Horizontal Expansion and enable Ironing I all switch off. When doing this in the Preferences/Setting Visibility window you realise just how many other setting options there are!

Now is a good time to save your new custom Profile. Go up to the Profile box where you selected the Extra Coarse profile and click the box. To the right of the box a star has now appeared as the Profile has been changed but not saved. In the drop down go down to,

'Create profile from current settings...', click. A 'Create Profile' name box appears. When naming Profiles I find it useful to give a descriptive name like 'Layer6_Nozzle2_Shell1'.

Alternatively you can click on the star and the 'Preferences' window appears and you can do your save there. Be careful not to overwrite the default 'Extra Coarse - 0.6mm' profile.

Infill

Infill Density - I do not use infill so set this to 0.

Infill Line Distance - this only becomes active if an infill is used. If an infill is used there are lots more settings to be explored.

Material

Wall Flow - this controls the speed of the screw/auger and as I have said it is linked to the nozzle size and the material diameter. The Flow can also be controlled later while printing on the LCD screen in the Tune setting under Flow. Keep Wall Flow at 100 % and edit once you find out how well your profile is printing.

Initial Layer Flow - keep this at 100 %.

Enable Retraction - I enable retraction.

Speed

Print Speed - the speed the Printhead moves while printing. With small printers I print between 25 - 30 mm/s. Large printers 40 - 60 mm/s.

Wall Speed - set to the same as the Print speed and then I deselect the visibility.

Travel Speed - the speed the machine moves from home or the top of the tower down to begin printing and the speed it moves between printing. Too fast and you can distort prints as the printhead moves between printing areas. I tend to set between 30 - 50 mm/s.

Initial Layer Speed - it is useful for the first layer to print slower to get going.

Enable Jerk Control - I do not activate this and in the Preferences untick the Visibility box.

Save your Profile settings by selecting 'Update profile with current settings' in the Profile drop down box.

Travel

Combing Mode - I leave this on 'All' not know whether it makes a difference.

Avoid Printed Parts When Traveling - I deactivate and hide.

Z Hop When Retracted - I activate this and when you do the height option appears.

Z Hop Height - This is good so the extrusion pulls up breaking the clay extrusion between printhead non printing moves. I use a setting of 1 - 2 mm.

Build Plate Adhesion

Build Plate Adhesion Type - I turn this off to None

Mesh Fixes

This group of setting I would suggest leave as are for the default Extra Coarse profile that we are basing our customised profile on. I am tempted to hide all these settings but if you have a 3D file that is not slicing well these are the settings you will use to try and improve the outcome.

Special Modes

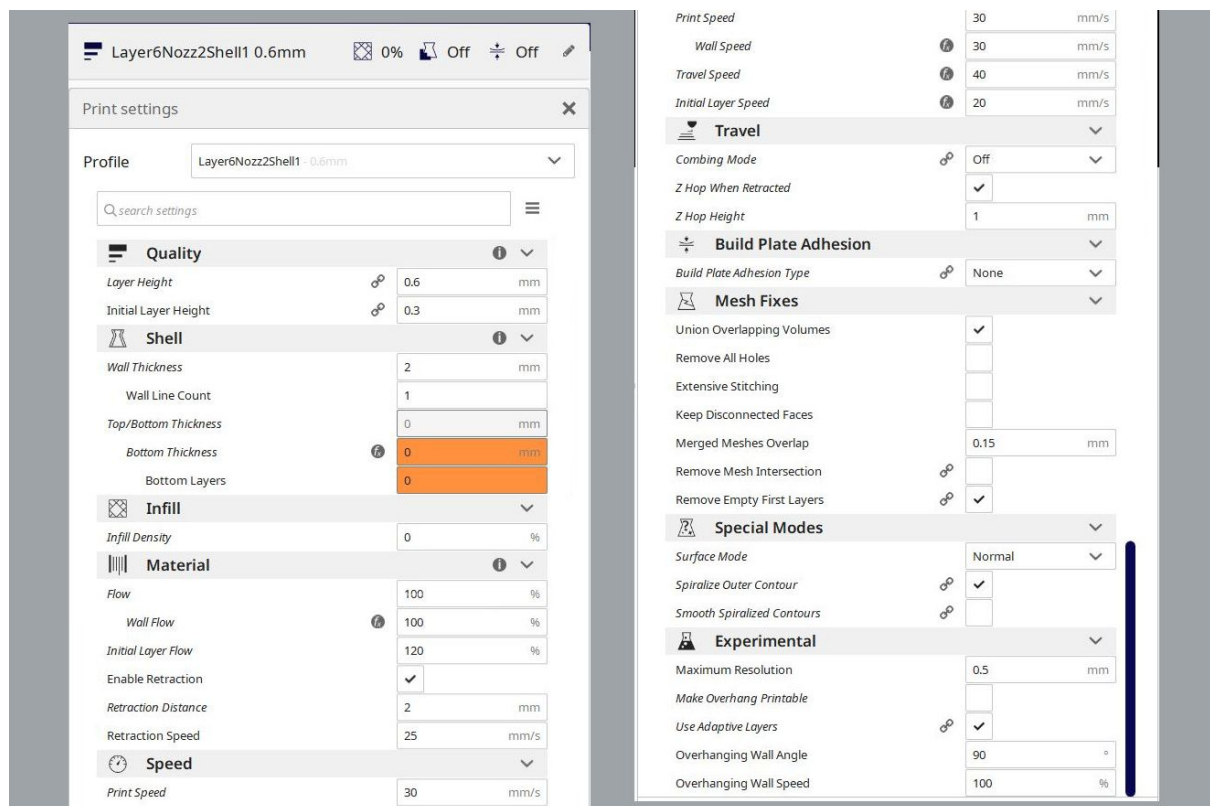
Surface Mode - this can be an important setting. A well drawn, watertight 3D file should slice perfectly in the 'Normal' mode and 'Normal' is the setting I save. However if you have a file where surfaces cross over or you want to print internal detail then the 'Surface' setting will try and slice all the surfaces in your model. I have not found the 'Both' option in the drop down to be of any use but try it on a problematic file.

Spiralize Outer Contour - I do select this but only if my print has a single shell or wall. It will not work on double or more walled prints and even on single wall objects you can often see the faint pattern of the Z rise - it is not always a perfect spiraling rise in the z.

Smooth Spirialized Contours - I leave this selected.

Experimental

This is another subcategory of setting that I keep as they are set in the default profile setting as I am not sure how well they work. An option like 'Make Overhang Printable' has to be tried now and then. If you have a look at the Preference window (cog icon in the subcategory navigation bar) you will find there are lots more Experimental Settings you could select and make visible to try.



Import Export Print Profiles

Any custom Profiles that you will have made can be seen in the Preferences window. Go 'Settings' (top left), in the drop down window go down to 'Profile' and in the next drop down go to 'Manage Profiles' at the bottom.

To Export, select the Profile you want and go 'Export'. (upper right)

To Import a Profile click 'Import' (upper right) and find the Profile file on your computer. It would appear you can import a .curaprofile file type that is new for Cura but also the old Cura Profile file type that was a .ini file type.

Cura Workflow

Once you have set up the Printer, the Material and made yourself a custom clay Profile and all have been selected and activated you are ready to slice a 3D model.

Click the 'Folder Icon' (top left) and find your 3D file.

The model is automatically placed on the build plate. Left click the model to select. All this is done in the 'Prepare' tab (center top)

Model Editing Tools

When a model is selected the tool palette on the left hand side of the main window becomes active.

Move - a model can be left click dragged, it can be numerically moved in the move window to the left or it can be moved on the x y z axis arrows. A model will only slice from the build plate up so the shape can be cut by moving the model below the build plate.

Scale - when selected left click drag will scale, the model can be scaled numerically, 'uniformly' or not depending on whether the box is checked and if unchecked it can be scaled on the x y z axis arrows.

Rotate - left click and drag on the x y z circles. There are also orientation buttons.

Mirror - hopefully self evident.

For the last two tools, Mesh Type and Support Blocker do a Google search. The mesh type can enable you to have different slice setting within one object.

Right Click Model Menu

This drop down menu is reasonably self explanatory. This is where you clear the build plate when you want to slice another model. The 'Reset All Model Positions' can be useful too.

Slice Button

Once you are happy with your model click 'Slice' or it might have sliced automatically. In the now familiar Preference window (go settings top left), left column , top, is a 'General' preference setting. In this you will see there is a selection box to 'Slice Automatically' or not. I prefer for it not to slice automatically otherwise each time you make a setting change or edit your model off it goes. Read through the 'General Preferences' as there could be settings from the default that you would prefer.

Slice Preview

Once slicing has taken place move onto the 'Preview' tab (top centre) or where the Slice button was, 'Preview'. There is a slider to the right of the main window to check the layers or a printhead animation controller along the bottom of the window. If you left click on either of these sliders you can then also use the keyboard arrows to move these animations. A left click mid window and then using the keyboard arrows moves the build plate. Middle mouse button held down will move the build plate and model in the window. Right click held down rotates and mouse wheel scales the visualisation - all the navigation you would expect in a virtual space.

Save gcode

The save button is bottom right once the slicing has taken place. If there is a removable disc in your computer this shows up as a preference to save to, but also offers an arrow drop with options. You can also go 'File' (top left), down to 'Export', select .gcode as the save as type in the window that opens and save.

Appendix - Detailed Research Documentation

1 - Testing of the relationship between Nozzle Size, Layer Height, Print Speed and Material Consistency for Clay Extrusion 3D Printing

Introduction

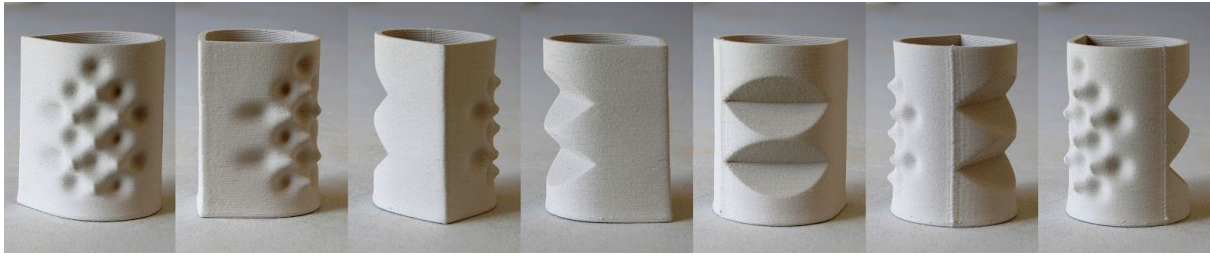
As with many technical processes there are a number of variables that go into successful clay extrusion 3D printing. Concentrating on print quality, these tests focus on the relationship between nozzle size, layer height, print speed, and material consistency . For convenience and the time individual prints take, the tests were kept to a small cup size scale using a standard shape of 8 cm height. While the specifics of these tests are relevant to this scale of 3D print it is hoped that the general conclusions and patterns of results will be relevant across all scales of clay extrusion 3D printing.

Previous tests have shown (see Clay Print Tests) that the physical properties of the material used, particularly the plasticity of the clay, has a significant influence on the print quality. To help concentrate on the specific objectives of this series of tests the variables of the material were standardised to a single clay body. A stoneware clay was selected (Sio2, PRAI 13 310 200) that was known to show good extrusion printing qualities in the printer being used.

Note: All images have been produced at high quality to enable zooming to see detail. (Ctrl+mouse wheel)

Benchmark Print

To set a benchmark for comparing print tests against, an initial sample of the shape was printed. From past experience a 1.6 mm nozzle was used with a 0.5mm slice height, print speed of 30mm per second and clay of a medium consistency. (See clay test for measurements of consistency) A double wall or shell was printed as this is known to offer stability to the print. It must be noted that all other tests were performed with a single wall or shell. The objective here was to prove that the sample shape could be printed crisply and without deformation, that was the case.

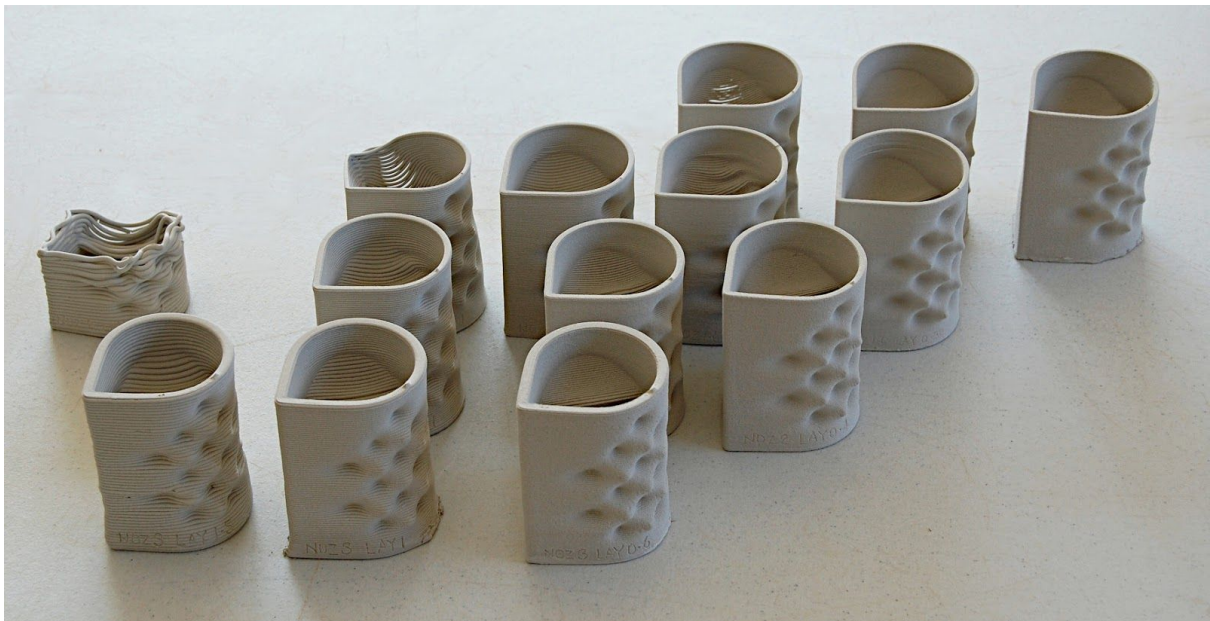


Benchmark print using a 1.6mm nozzle, layer height 0.5mm, medium consistency clay and double wall or shell.

Nozzle to Layer Height Tests

The first objective was to undertake a series of tests to see if an ideal ratio could be found between nozzle size and layer height for successful prints. The print speed and material consistency were kept constant. Clay of a medium consistency was used at a print speed of 25mm per second.

Because of the huge variation in the character of clays, to talk of ideals in clay printing is dangerous. So it would be better to say the objective was to find the general sweet spot for the proportion of the diameter of the print nozzle and the sliced height of each print layer. Four nozzle sizes were tested, (1mm, 1.6mm, 2 mm, 3 mm) with the overall pattern of proportion being important rather than individual tests.



Overview of tests - nozzle size decreasing from front to back and layer height decreasing left to right.

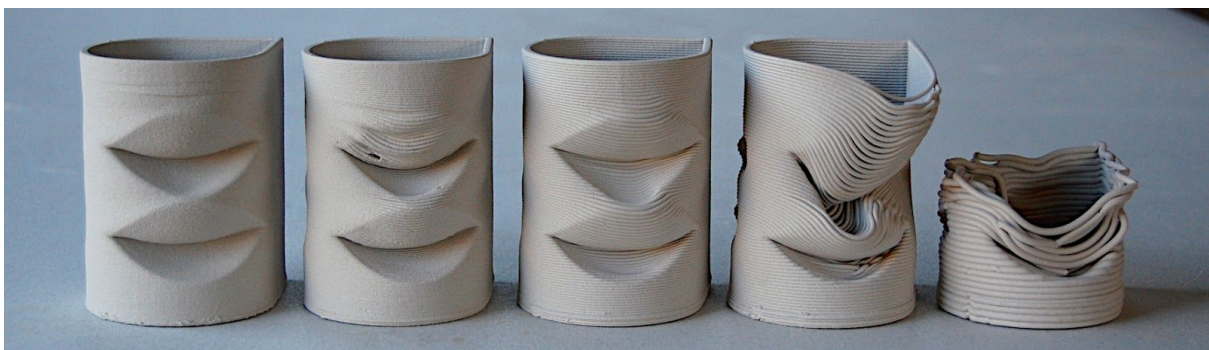


Nozzle size - (left to right) 3 mm, 2 mm, 1.6 mm and 1 mm.

Procedure

To begin with an initial set of tests was undertaken to establish the most likely proportions for further testing. A test of 1:1 (nozzle diameter:layer height in measurement) proved to break down quite soon after reaching the textured area and overhang in the test shape - extreme right in the photo below. Using a 2 mm nozzle with a 2 mm layer height, the printed layer sunk below the printhead nozzle height and the connection between print and printhead was lost.

Next a 1.6 mm nozzle with a 1 mm layer height was tested, offering a ratio close to a 3:2 proportion - second from right in the photo below. While this printed better than the 1:1, the layers were not adequately packed together to cope with the overhang. This proportion printed fine in the vertical and coped adequately without distortion or delayering in the textured area, but collapsed very early in the 45° angle.



Tests determining nozzle to layer height proportions worth considering. Left to right - 5:1, 3:1, 2:1, 3:2, 1:1.



Proportions of 2:1, 3:1 and 5:1 all proved to print successfully so a test was designed using these three proportions across the four nozzle sizes.



The tests, as laid out in the illustrations below, are designed first to compare the crispness or sharpness of a printed corner for each of the four nozzle sizes. Each nozzle size is tested making use of the three nozzle, layer proportion ratios. Secondly, designed into the test form is an area of low relief pattern so as to be able to compare how nozzle and layer height affects the print quality of such a repeat pattern. Thirdly, there are two 45° angled indentations to the form, to be able to observe how well the different settings cope with the problem of clay slumping on an overhang during printing. Bearing in mind that the two walled benchmark test printed showing no sign of slumping.

All tests were printed on a small Delta type printer (CERAMBOT) and as said using clay of the same medium consistency and at the same print speed of 25 mm per second. Compressed air was used to deliver the clay to the printhead, with more air pressure required to feed adequate clay flow for the larger nozzle diameters. The printhead screw speed, having previously been configured in the slicing software (Cura) will have automatically changed with each new gcode file that was prepared. Each test was printed with a three layered base.



Note: All photographs have been embedded in this document at high quality so as to be able to be zoomed for detailed observation.


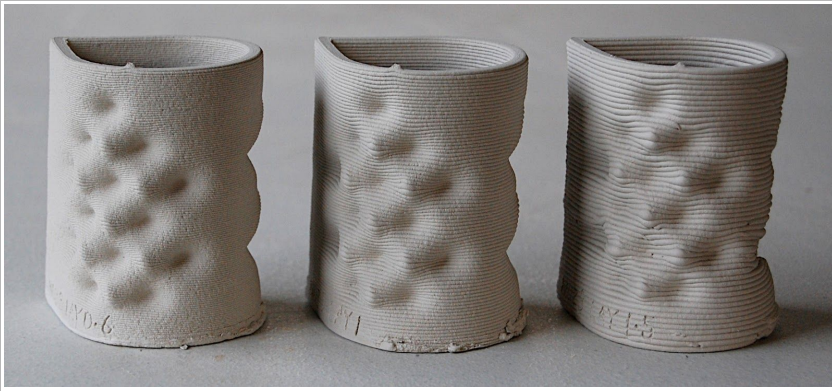
Vertical Edge Test

Nozzle : Layer	1:5	1:3	1:2
Nozzle 1 mm			
Layer height	0.2 mm	0.33 mm	0.5 mm
Nozzle 1.6 mm			
Layer height	0.32 mm	0.53 mm	0.8 mm

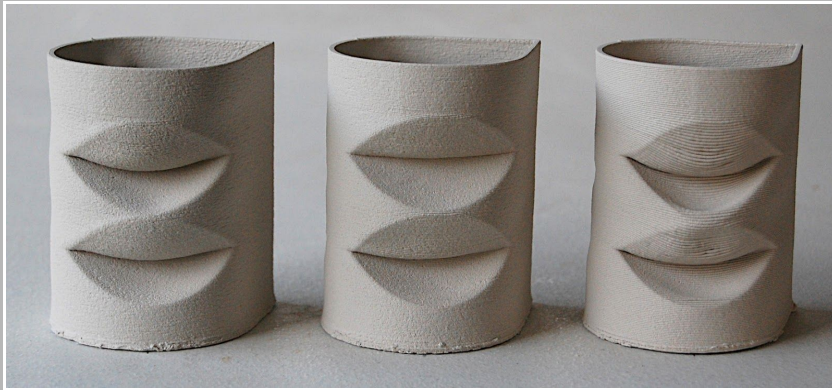

Nozzle 2 mm			
Layer height	0.4 mm	0.6 mm	1 mm
Nozzle 3 mm			
Layer height	0.6 mm	1 mm	1.5 mm


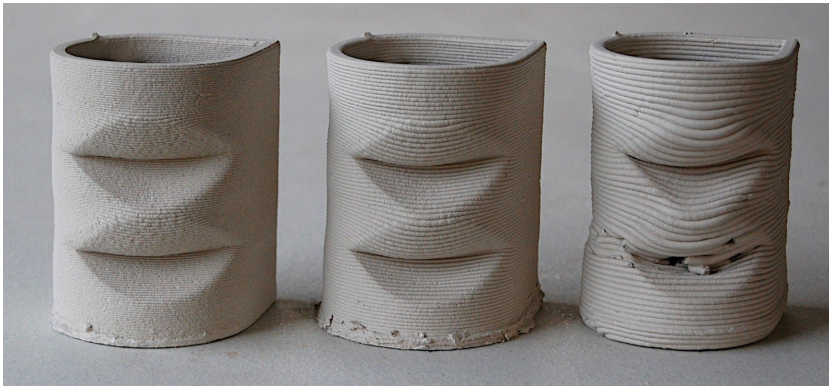
Low relief Pattern Test

Nozzle : Layer	1:5	1:3	1:2
Nozzle 1 mm			
Layer height	0.2 mm	0.33 mm	0.5 mm
Nozzle 1.6 mm			
Layer height	0.32 mm	0.53 mm	0.8 mm

Nozzle 2 mm			
Layer height	0.4 mm	0.6 mm	1 mm
Nozzle 3 mm			
Layer height	0.6 mm	1 mm	1.5 mm

Overhang Test

Nozzle : Layer	1:5	1:3	1:2
Nozzle 1 mm			
Layer height	0.2 mm	0.33 mm	0.5 mm
Nozzle 1.6 mm			
Layer height	0.32 mm	0.53 mm	0.8 mm

Nozzle 2 mm			
Layer height	0.4 mm	0.6 mm	1 mm
Nozzle 3 mm			
Layer height	0.6 mm	1 mm	1.5 mm

Speed Test

The purpose of this test was to determine whether the print speed has any significant outcome to the final result. For this test a 2 mm nozzle was used throughout, as was a clay of medium consistency and a layer height of 0.6 mm. A three layer base was again included in each print.

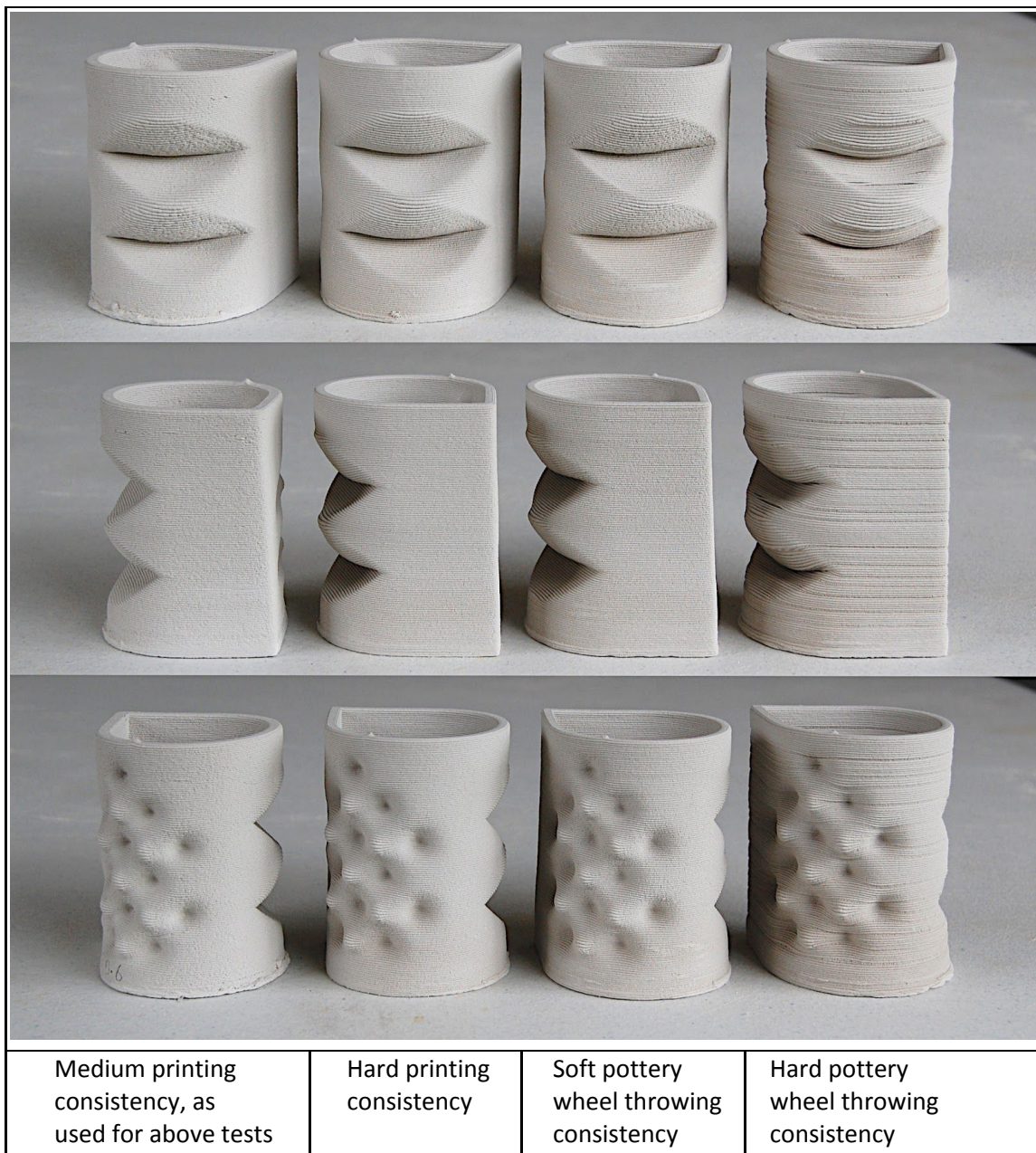
It must be noted that the printer used is stable and smooth at both low speed and higher speeds so there were no inaccuracies due to mechanical limitations. Further, for the purposes of a fair test it was ensured that the clay delivery was generous at all speeds and there were no shortcomings due to material feed. A large difference in print time, from 42 minutes to 12 minutes was designed into the test to ensure greatest comparison.



Stiff to Hard Clay Test

With clay extrusion printing there is a logic that the stiffer the consistency of clay used the more stable the print should be. Stiffer, or the harder the clay is, the more the print should stand up and not distort or collapse.

Two tests were designed to test this speculation. The first again makes use of the test shape as used above and prints were made using a 2 mm nozzle with a layer height of 0.6 mm and a print speed of 25 mm per second. The second group of tests makes use of a flared shape with a 45° angle to it and a base of 4 cm, a height of 6 cm and a rim diameter of 16 cm. These tests were made with a 3 mm nozzle and a layer height of 0.6 mm and printed at 25 mm per second.



For these tests four samples of clay were prepared from a medium to hard consistency. As a rough visual guide, the clay of a medium printing consistency makes a soft ball in the hands and the clay sticks and soon builds up on the hands as handled. A ball of hard printing clay is still soft to the touch but if your hands are clean can be handled with only a small amount of clay sticking to the hands. Handling of soft pottery wheel throwing clay, little sticks to the hands and a ball is reasonably resistant to handling. Hard throwing clay, your hands remain clean and some force is required to knock up a ball.

	Medium Printing consistency	Hard Printing consistency	Soft Pottery Wheel Throwing consistency	Hard Pottery Wheel Throwing consistency
Drop Spike - this tool represents the distance a 38 cm (235gm) sharpened steel rod penetrates	30 mm	26 mm	22 mm	17 mm

into the clay sample when dropped from a constant height of 9 cm.				
The pressure used during printing. For the hardest clay a mechanical ram clay delivery was used.	2.5 Bar	4 Bar	6 Bar	Mechanical ram clay delivery used.



Clay Stiffness Test - hardest clay to the left softest to the right.

For the flared shape test the objective was to run the test until the print broke down. To ensure the test was printable a benchmark was again set - right in photo. This was once more printed with a double wall using the medium printing clay, a 1.6 mm nozzle, layer height 0.5 and at 30mm per second. Also the hard printing clay (Hard Pottery Wheel Throwing consistency) was not tested.

In the illustration the 'Soft Pottery Wheel Throwing' consistency clay test is to the left. Followed by the 'Hard Printing' consistency clay, second left. The two right hand tests are the same 'Medium Printing' clay. The extreme right hand sample is the benchmark. The second to right, as with the first two tests are single shells prints using the 3 mm nozzle.

Conclusions

There is no desire to see these tests as providing hard and fast rules but to be a documentation in time that offers a reference enabling informed judgments and decisions to be based on. With the variability of clay and the complexity of the technology used, the aim was to offer some insight into the general pattern of the processes involved rather than the specifics.

Nozzle to layer height proportion

The decision of what nozzle size to layer height proportion to use when extrusion printing will always be an individual choice based on personal preference and the type of work or job being done. What this research has shown is that there are workable parameters between a proportion of 1:2 up to 1:5 layer height to nozzle diameter.

From the nozzles tested it appears 1:3 is a good starting point. A further observation is that as the nozzle gets larger the ratio proportion moves towards the smaller end of the ratio scale. For example a ratio of 1:2 produces a clean print with a 1mm nozzle while a 1:5 ratio might be preferable with a 3mm nozzle.

On the vertical edge test it stands to reason that the finer or smaller the nozzle the sharper or crisper a corner will be - a corner can only be as sharp as the diameter of the clay extrusion. The layer height appears to have little significant effect on the edge quality.

Layer height did show to have more significance in the low relief pattern test. All printed adequately so choice will come down to personal preference with the larger proportion being strongly layered and smaller proportions becoming granular in texture as layers become compacted together. This becomes more pronounced as the nozzle size gets smaller.

The 45° degree indentation test did not have as marked differences as expected. There is certainly a balance to be had where the contour stepping happens on a 45° slope. Too much flattening that happens with a fine proportion setting will distort the print while too great a contour step with a large ratio proportion results in the layer breaking apart. Possibly for this reason the mid range of 1:3 often proved to print with the least distortion.

An observation made while printing, that is obvious but worth bearing in mind, is that the larger the nozzle the more stable the print. While the proportion of the height to width of the extrusion was being kept constant for each nozzle size, as the wall got wider with each nozzle size, the print looked more stable during the print. That said, while the printed wall looked very delicate and wavy or unstable during printing of the 1mm nozzle the prints turned out fine.

Print speed

While the speed test outcomes do not look very interesting the test was very revealing for this very reason. To watch a printer print at 40 mm per second, taking 12 minutes to print the sample shape including a three layered base. Then to watch the same shape being printed at 10 mm per second and taking 42 minutes and there then being very little difference between the two prints is quite striking.

It would appear that as long as the equipment can stand up to it, printing speed is not that important in the outcome of print quality. Obviously this is assuming that you have a fully balanced system where clay delivery and mechanical smoothness can take the speed.

Stiff to Hard Clay

The two tests designed to examine the effect of progressively stiffer clay consistency being used on printed samples were both most informative and in different ways.

On the upright sample shaped, it showed that there is very little difference to print quality between a soft and hard clay consistency. The repeat pattern printed very similarly across all four tests. There are variable flow rate layering issues on the test using the hardest clay but these are due to the equipment and not the clay. Because of how stiff the clay was, a mechanical ram was used to feed the clay to the printer for this test and the variation in surface quality was due to not controlling the ram speed well. It was expected that the harder clay would show less distortion on the 45° degree indentation but as can be seen in the photographs there is little difference across all four test samples. The stiffer clay stood up no better than the clay of medium consistency.

The photographic documentation of the flared sample shape tests tells its own story. The harder the clay the less the flare will build. Contrary to expectation the stiff clay does not stand up in this shape better than a medium softer clay. It would appear the stiffness of the extrusion collapses the wall underneath. On a vertical build this would not be a problem but with any shape that goes off the vertical it would appear that there is no advantage in using stiffer clay. It must be pointed out that as has been shown from other tests (see Formulating and Testing a Clay Body for Extrusion 3D Printing) that overly soft clay will also not work.

Optimising any system is often by small degrees and this series of tests has gone some way to offering insights into where clay extrusion printing can be nudged. The information on nozzle, layer ratio probably only confirms what practitioners have arrived at through trial and error. People will print at whatever speed they and their equipment are comfortable with but it is reassuring that whatever speed that is there is no great advantage in changing it. The lack of any discernible advantage of using stiff clay, except for vertical printing, possibly has the most relevance. Using stiff or hard clay requires more robust equipment and if there is no advantage to be gained then there is not the requirement to explore the increased engineering challenge.

Telling is that the best result obtained where those of the benchmark tests that were arrived at through gradually attained experience, using standard but well configured equipment and careful technique.

2 - Testing of Six Clays for Extrusion 3D Printing

Introduction

Clay as a natural material is not uniform across all its sources, offering different characteristics from deposit to deposit and type to type. For extrusion 3D printing clay can be used as dug, after cleaning and preparation or be a composite of many clays and or other additive materials such as grog and fluxing materials. The aim of this research was to give some practical measurements to what characteristics are desirable for clay 3D printing.

The approach taken was to mix six different clays to the same consistency and to then undertake the same tests on each sample. It is the rheology, the character, the deformation and flow of the clay mixed as a paste that was to be studied. The clay consistency was judged initially by eye and feel, as this is how clay for printing has generally been evaluated.

Note: All images have been produced at high quality to enable zooming to see detail. (Ctrl + mouse wheel)

Clays Tested

China Clay (Kaolin Clay) - Grolleg China Clay, mined in Cornwall, UK was used. The sample was mixed from powder clay. Too much water was added so the mixture was left overnight to dry out until the desired standard consistency was reached.

Porcelain Clay - this was a recycled mix of a number of commercially supplied porcelain bodies. The sample had been stored at the desired paste consistency in plastic.

Ball Clay - an off white secondary plastic clay, Hyplas 71 was used- mined in Devon, UK . Mixed from a powdered state, it was mixed softer than required to ensure thorough wetting and left to dry overnight to the desired consistency.

Fine Stoneware Clay - a clay I like to use, 164 ARTE/PRAI White Stoneware Clay from Spanish supplier Sio-2 that has 40% fine grog (0 - 0.2 mm). Water was added to clay of throwing consistency and mixed to a paste.

Coarse Stoneware Clay - Ivanhoe, supplied by Potclays UK, a clay I have used for large 3D printed objects. Buff in colour it has 20% grog sieved to 40# grade (0 - 0.5 mm).

Red Clay - a mixture of recycled red clay that is sandy but fine and smooth. Stored in a soft consistency.

Clay Paste Consistency

Each sample was thoroughly mixed by hand so each felt and looked to be of the same consistency. This consistency is not so soft that the clay looks wet on the surface but also not so hard that when handfuls are slapped into a stack that there is not some give to each layer. With clean, dried hands the balls of clay can just be handled without too much clay sticking to the hands but a stickiness will build up on the hands as a ball is knocked into shape - the clay paste is soft, but just stiff enough to be handled.



China Clay

Porcelain

Ball Clay

Fine Stoneware

Coarse Stoneware

Red Clay

Test Procedure and Equipment

There were a number of aims to this research -

- To assess the clay to water ratio in clay mixes of the same paste consistency.
- To develop a simple tool to give measurements to soft clay consistency.
- To measure the pressure or force required to extrude soft clay of the same consistency through a narrow nozzle.

- To measure the comparisons of the flow rate of the different clay samples in a 40 cm long narrow tube.
- To sample the 3D printing characteristics of each clay.
- To measure the shrinkage of the different clays when dry, when fired at 1000° c and 1220° c.

Water Ratio

A 200gm sample of each clay of the same consistency was dried out and reweighed when bone dry, as dry as possible in a domestic atmosphere. From this the water content of the paste could be calculated.

Clay Consistency - Drop Spike

A simple dropping spike tool was made that has proved to be reasonably consistent and accurate. Dropped onto the clay sample, down a guide tube from a constant height, the length of penetration would be averaged over four attempts and recorded. To my surprise the six clay samples, mixed to be adjudged the same consistency by hand and eye were all within 5% of each other on the 'Spike' measurement. See [Drop Spike](#) document.



Syringe Pressure Extrusion

A 60ml syringe was adapted to have a removable end plate and nozzle. An 8mm nozzle was fitted so not too great a pressure was required to judge when extrusion or flow was attained. The extrusion force was measured by pressing the syringe plunger down onto a bathroom scale. This test was more specifically to aid an understanding of clay rheology for printers where the clay bulk is carried near or on the printhead.



Tube Pressure and Flow Rate

A 500ml container was filled with each of the clay samples, pressurised and the flow rate from an attached 40cm long, 6mm inside diameter plastic tube was measured. This test offers quite different results from the Syringe Extrusion results above and is designed to aid those printers where the clay bulk is carried alongside the printer and the clay paste is forced through a length of tube to the printhead. Clear comparisons of the flow rate across all six clays could not be gained as the Coarse Stoneware and Red Clay required more pressure to reach the 40cm tube length. The Chine Clay, Porcelain, Ball Clay and Fine Stoneware Clay were all tested at a pressure of 3 bar (43 psi) while the Coarse Stoneware and Red Clay were tested at 4 bar (58 psi) of pressure.



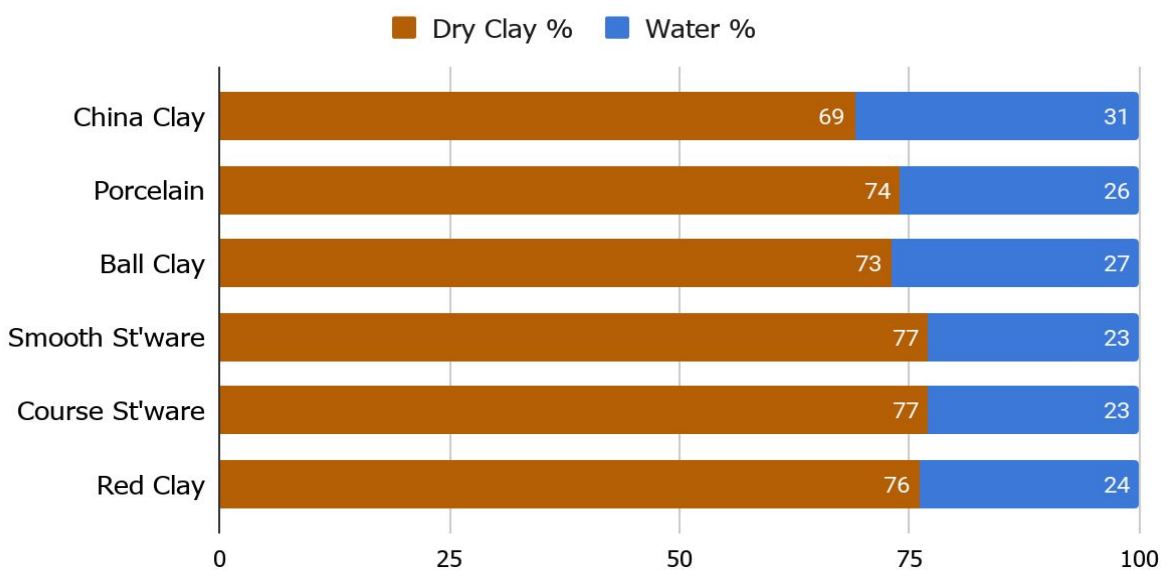
3D Print Quality

A small Delta type printer with a 40cm clay delivery tube was used to make the print samples. A simple 6cm diameter and 6cm high cylinder was selected as a print sample shape. A spiralized gcode was prepared in the Cura slicing software using a 2mm diameter printhead nozzle, a 0.7mm layer height, at a print speed of 25mm per second. The same print file was used for each clay sample and not surprising each print came out very similar. The clay delivery pressure just had to be adjusted in line with the results obtained from the Tube Pressure Flow Rate tests.

Test Results

While the composition, character and rheology of clays is renowned for being difficult to pin down and there are no claims for the testing equipment to be highly accurate, in the overall patterns of these results there is useful information to be found for making informed choices. This information gives a sense of what to expect when choosing clays to use for extrusion clay 3D printing.

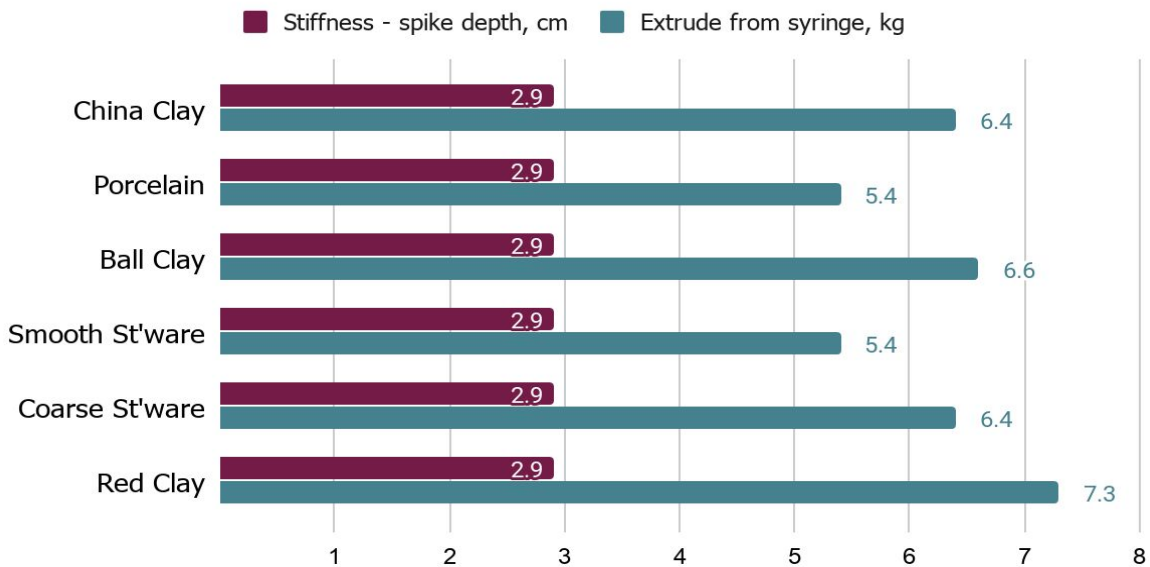
Clay | Water



These results clearly show that different clay of the same consistency, measured by feel and eye and confirmed by the spike drop measurements have different ratios of dry material to water. This has relevance to clay extrusion 3D printing, as for the stability of complex shapes with overhangs it could be useful for the clay to dry quickly - the less water the quicker the clay dries. Other printing techniques might require slow drying clay. In the tests the pure clay, China clay and Ball clay required most water. It was expected that the water content of the Red clay would have been higher but it is not known how much non clay material, in the form of fine sand there is in the clay tested. Blended clay bodies such as the Porcelain and the two Stoneware bodies that are made up of a percentage of non clay materials were expected to have less water content. The shape of clay particles are flat or platelet-like, taking up more water between the layers, than the more rounded,

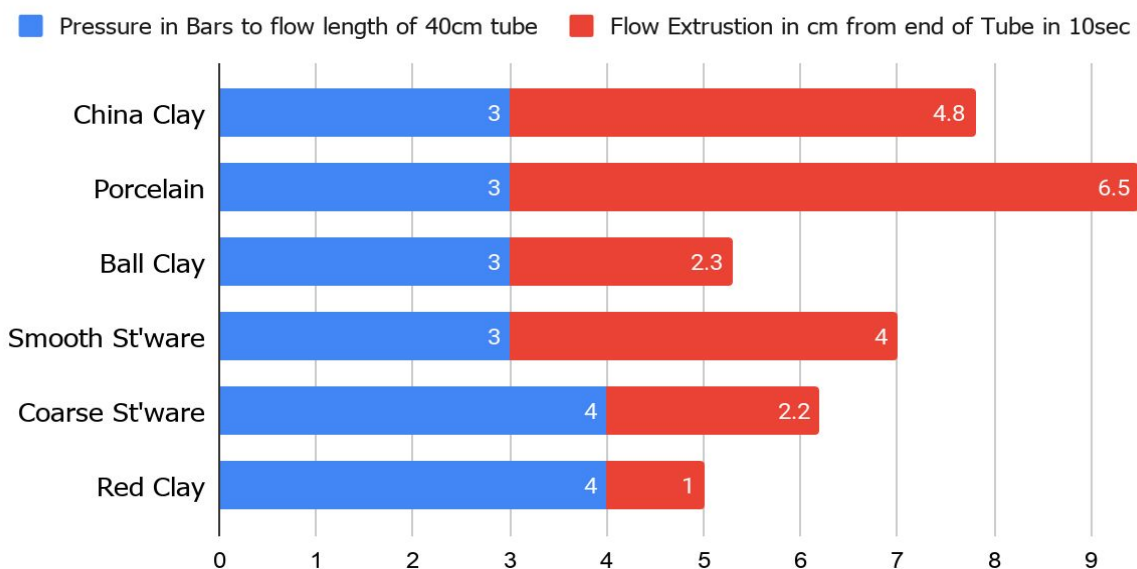
non clay particles of grog, that is a crushed fired ceramic material or crushed feldspar or silicate rock.

Syringe Pressure Extrusion



The aim of this test was to obtain measurements illustrating the variation in force required to extrude different clays of the same consistency through a short wide nozzle. Each clay was mixed to have a spike depth measurement of 2.9 cm. Single material, pure clay, homogenous clays (china clay, ball clay) are shown to require more force than blended clay mixes (porcelaine, stoneware mixes). Also as a general observation the darker the natural colour of the clay the more force was required, with the Red Clay requiring the most force. Text books will also show that these are the more plastic clays. Clay bodies with non clay material in the composition, for example Porcelain with 50% non clay (feldspar, flint) and the Smooth stoneware body with 40% non clay (grog) material extruded under the least force.

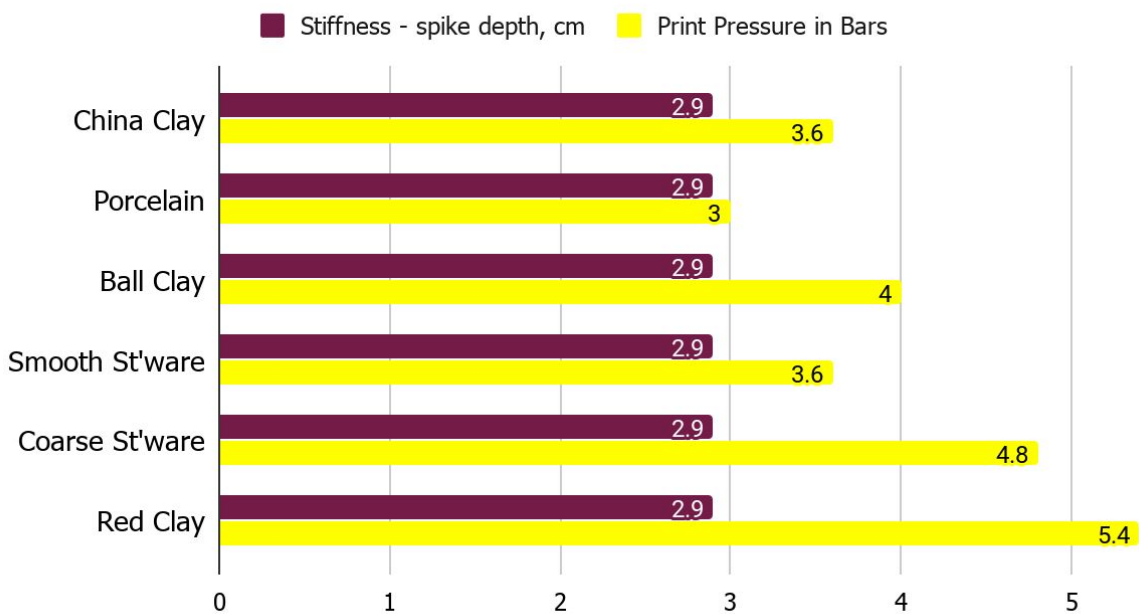
Tube Pressure and Flow Rate



These results make interesting reading alongside the Extrusion Force measurements above. The objective was to gain measurements for the rate of extrusion flow for the different clay samples

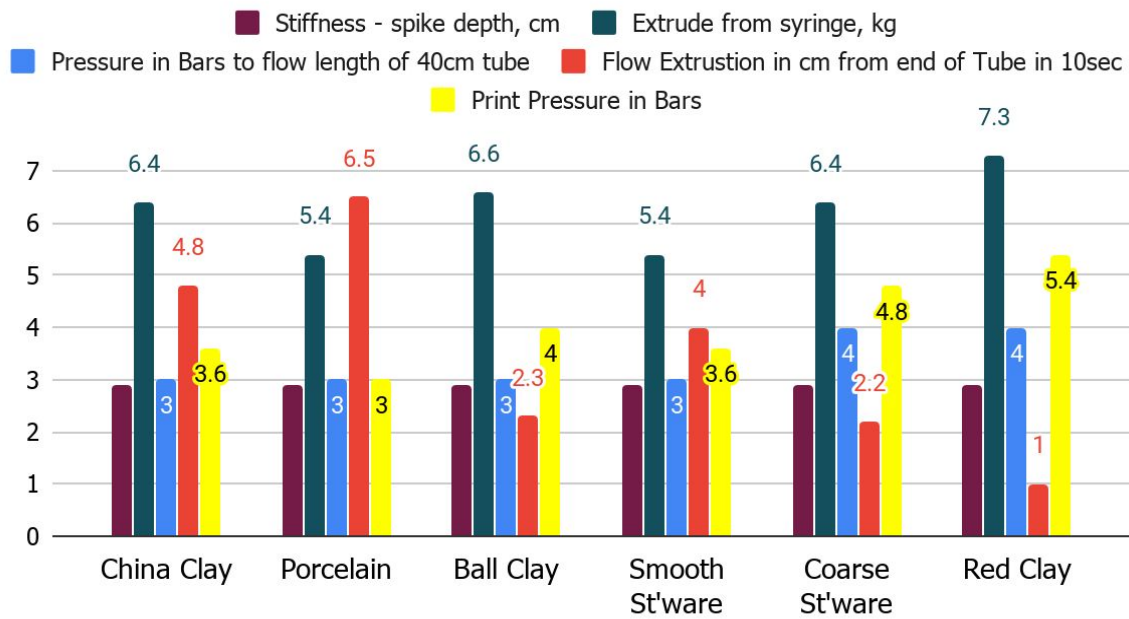
having traveled through a length of narrow tube. In the table above the last two clays required more pressure, up to 4 bar, from 3 bar of the other four clays to travel the length of the tube. The results of flow for the Coarse stoneware and Red clay are in fact more restrictive than the graph visualisation indicates. The general pattern of Extrusion Flow is consistent with the Extrusion Force tests. However there are interesting anomalies. For a short extrusion from the syringe, the force required to extrude China clay and the Coarse stoneware were similar. The results in the tube test were very different. China clay flowed out of the 40 cm long tube at 4.8 cm in 10 seconds under a pressure of 3 bar. The Coarse stoneware required 4 bar of pressure to be able to test a flow from the end of the tube and then the flow was only 2.2 cm per 10 seconds. Another interesting comparison is between the Ball clay and Coarse stoneware. The Ball clay needed slightly more force than the Coarse stoneware on the Syringe extrusion test. In the Tube flow test the Coarse stoneware required the 4 bar of pressure to flow the length of the tube and then resulted in a flow speed only slightly less than the Ball clay that was only under 3 bar of pressure. Similarly, comparing the Porcelain and Smooth stoneware clays, where both registered the same Syringe extrusion force. In the Tube test the Porcelain flowed notably faster than the Smooth stoneware under the same pressure. It is clear that clay behaves differently when under pressure and extruding a short distance and when being forced down a long tube.

Clay Stiffness and Print Pressure



It must be pointed out that these results are for a Delta type clay printer where the bulk clay is delivered down a 40 cm tube to the printhead. For printer designs where the bulk clay is delivered close to the printhead, then the results would be expected to be more consistent with the Syringe Pressure table. These results are more in line with the Tube Pressure Flow Rate table. With the clay in the printer bulk container all of the same consistency it shows how different clays require different amounts of force to obtain a similar, constant clay feed to the printhead.

Stiffness, Syringe Force, Tube Pressure, Tube Flow, Printing Pressure



Clay Shrinkage

To obtain shrinkage measurements the printed cylinder samples are measured against the dimensions of the original digital model. Measurements were taken once dry, after a bisque firing of 1000°C and again after a low stoneware firing of 1220°C.



The six clay sample cylinder prints from left to right - China Clay, Porcelaine, Ball Clay, Smooth Stoneware,

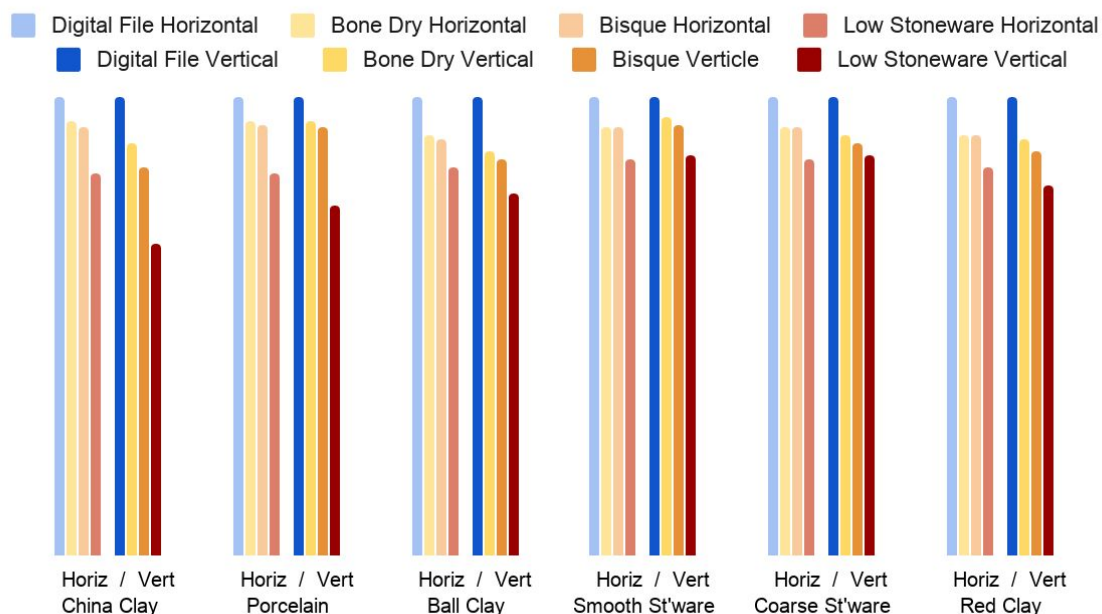
Coarse Stoneware, Sandy Red Clay. Top row - bone dry. Middle row - fired to 1000° c. Bottom row - refired to 1220° c.

The horizontal diameter of the inside of the cylinder was measured and the vertical height. The aim was to record the different shrinkage rates of each clay but also the difference in shrinkage of the height and width of the printed samples. While clay shrinks uniformly it has been observed that clay 3D printed objects do not. It is speculated that this is probably due to the vertical pressure of the layered extrusion and the clay memory.

Comparison of the photographic reference from bone dry to bisquit, to a low stoneware firing makes for interesting reading. The china clay that has been shown to have the highest water content at the time of printing did not shrink the most at bone dry. Ball clay with the second highest water content has clearly shrunk the most. It clearly shows that there is more to the properties of shrinkage in clay than the loss of water up to the bone dry state.

The ratio of the shrinkage of the samples at the different states (dry, bisque, high fire) is also not consistent over the range of clays. Looking along the line of tests in each state, it can clearly be seen that in the line of the highest firing there is a distinct pattern to the height shrinkage from left to right, that is not the case when bone dry or bisque fired. China clay has shrunk the most, stepping up towards the Coarse Stoneware that has shrunk the least and then back down to the Sandy Red. The ratio of shrinkage of the ball clay has remained reasonably constant throughout each state while the Porcelain clay has shrunk much more from bisque to high fired. It becomes clear that different clay decisions will be made for different final temperatures.

Clay Shrinkage

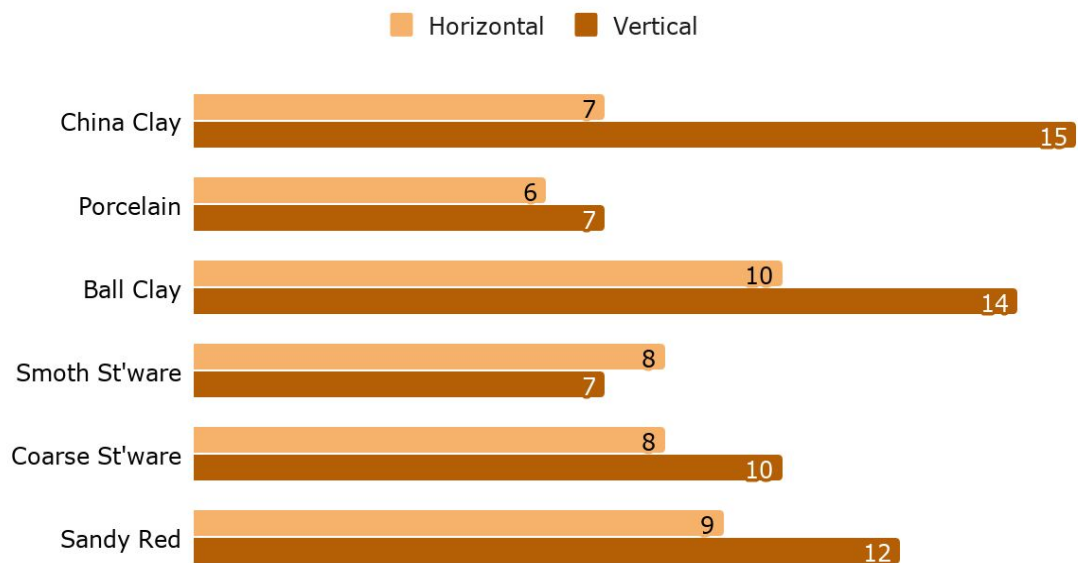


If there is a general pattern to be observed, it is that the pure clays, such as the china clay and the ball clay shrink the most. Clay bodies with non clay material such as grog and sand, as in the two stoneware bodies and the sandy red clay shrink the least.

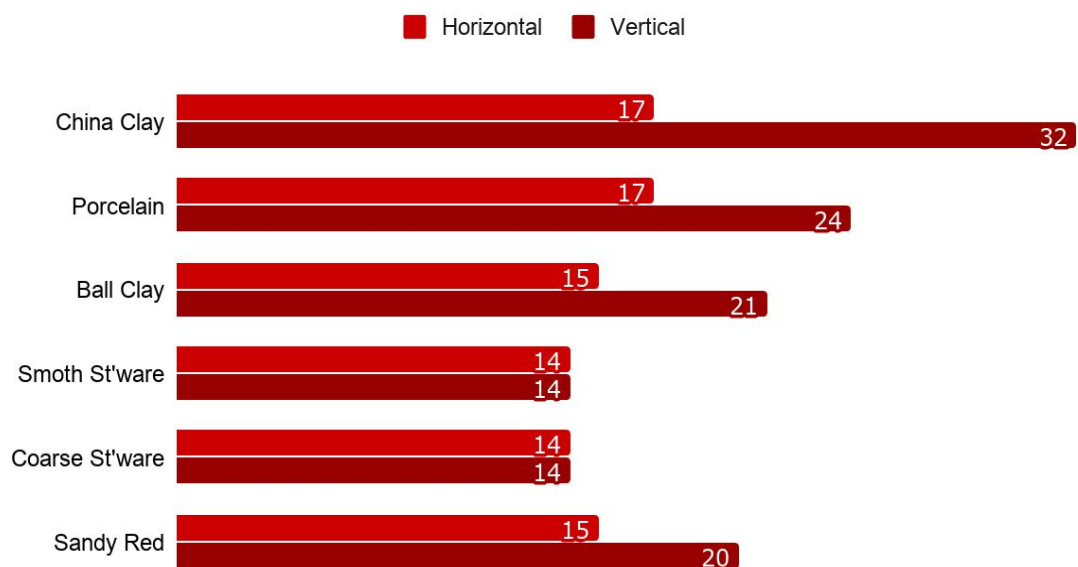
Possibly the most interesting information to come from these tests is the data on the difference between the horizontal and vertical shrinkage of these samples. Once again clear patterns of shrinkage are difficult to pinpoint due to the different makeup of the clay bodies tested. Fired to a bisque temperature of 1000°C there is a clear discrepancy in horizontal and vertical shrinkage with china clay shrinking nearly double in the vertical to the horizontal. Fired to a low stoneware temperature of 1220°C there is still this discrepancy with china clay but interestingly the smooth and coarse stoneware clay have now equaled out in shrinkage at 14%. If there is a

pattern, then as the firing temperature increases the discrepancy reduces in clay bodies with high non clay material.

Shrinkage at 1000°C as %



Shrinkage at 1220°C as %



3 - Formulating and Testing a Clay Body for Extrusion Clay 3D Printing

Introduction

While almost any clay can be used for clay extrusion 3D printing it is clear some clays print better than others. This variation can be dependent on the characteristics of the clay but also the type of machine being used and the style of work being produced. The aim of this research was to gain an insight into what considerations there are when formulating a basic clay body recipe from processed powdered raw ingredients. As available ceramic materials will vary from region to region, it is the process of formulating the clay body and the outcome of the tests that are important rather than the actual recipe, although the recipe could act as an initial guide for further development. This initial clay recipe was tested to evaluate its suitability. The effect of clay consistency, in the form of soft, medium and hard clay on the print outcome was tested. The tests developed onto considering what advantage clay additives might offer to this basic recipe and what improvements could be made to the initial recipe.

Note: All images have been produced at high quality to enable zooming to see detail. (Ctrl+mouse wheel)

The Clay Recipe

This recipe was formulated for a printer where the clay bulk is situated alongside the machine and the clay is piped to the printhead via a length of tube. It was also designed to be stable at high temperature. As a starting point the recipe was based on a plastic ball clay to offer plasticity and a smooth extrusion. However, plastic clays tend to have higher surface tension so are more difficult to force through narrow tubing. China clay was introduced as tests have shown china clay requires less pressure to be extruded and has a better flow rate through tubing. Fine grog was also included as non clay materials have also been shown to reduce clay body surface tension, thereby increasing flow rate and reducing the extrusion pressure. Further, a proportion of grog in a clay body is known to lower shrinkage, aid in drying and offer a range of particle size that should help with plastic stability. The choice of grog particle size needs to be in keeping with the printer nozzle size to avoid blockages.

Test Mix 1

Ball Clay (Hyplas 71)	44
China Clay (Grolleg)	33
Grog (Fine Molochite)	22

Hyplas 71 Ball Clay	A Devon (UK) ball clay with low iron content, high plasticity. Fires ivory buff colour.
Grolleg China Clay	A blended English china clay, combining moderate plasticity, used in white earthenware, bone china and porcelain bodies. White firing.
Molochite Grog	Manufactured using calcined china clay this is a refractory white grog. A 200 mesh fine grade was used.

While the recipe was given numerical values in percentage by weight a note was made of each material by volume. While proportionally less in weight the china clay in volume was far greater than the ball clay. Considering the idea was to add china clay, for its physical properties to a predominantly plastic ball clay body this turned out not to be the case. In volume this body has more china clay and proved not to be very plastic and 'short' in character - when printing the extrusion broke very easily. On checking the 'formula weight' of Grolleg china clay (276) and that of Hyplas 71 ball clay (541) this difference in weight to volume is explained.

Ball Clay (Hyplas 71)	800 gm	880 ml
China Clay (Grolleg)	600 gm	1180 ml
Fine Grog (Molochite)	400 gm	400 ml

Water to Dry Ingredients Proportions

To be able to measure the amount of water in the soft, medium and hard consistency mixes, the tests were prepared from dry ingredients. Left for a day before testing to allow for the water to be absorbed into the clay this short time of 'wetting' of the clay can further account for the 'short' non-plastic character of the clay tests.

A bulk mix of dry ingredients, ball clay, china clay and molochite grog was mixed and dry sieved twice to ensure an even mix. Test samples were prepared from this bulk mix in proportions where dry ingredients and water together total 100%.

	Dry Ingredients %	Water %
Soft Mix	69	31
Medium Mix	72	28
Hard Mix	75	25

Samples were mixed and left to saturate for the 24 hours as mentioned and then further mixed. The hard clay sample could be formed into a ball in the hands with little clay sticking to the hands. Handling the medium sample, sticky clay would progressively build up on the hands but a ball could easily be formed and handled. The soft mix could be formed into a ball in the hands but it was a sticky and messy activity.

Measurements of Clay Consistency

	Soft Mix	Medium Mix	Hard Mix
Dry ingredients: water by weight.	69:31	72:28	75:25
Drop Spike - this tool represents the distance a 38 cm (235gm) sharpened steel rod penetrates into the clay sample when dropped from a constant height of 9 cm. See Drop Spike document.	54 mm	32 mm	22 mm
Syringe Extrusion - this tool consists of an adapted 60ml syringe to have an 8 mm nozzle. The measurement represents the force required	4.1 kg	6.3 kg	10.2 kg

to extrude the clay sample through the 8 mm nozzle constriction that is 30 mm in length.			
The pressure required to force the clay sample the length of a 40 cm tube with an inside diameter of 8 mm.	2 Bar	2.5 Bar	3 Bar
The pressure used during printing.	2.5 Bar	3 Bar	5 Bar

Printing Tests

Two shape print files were prepared to be printed and the tests were run on a small Delta printer (CERAMBOT). One shape file was a standard 6cm high and 6 cm diameter cylinder. The second is an 8 cm height cylinder with a diameter of 6 cm that has three shape tests designed into it. First a vertical corner, second a repeat relief texture and thirdly two 45 degree indentations. In all the following tests the same .gcode file for each of the two shapes was used.

Cylinders

The 6 cm cylinder shape printed much the same for each of the three clay consistencies with more air pressure being required as the clay got stiffer - see table above. A 2 mm nozzle was used throughout and there was little if any noticeable variation in printed wall thickness across all three consistencies. However there are two points of interest worth noting. One is the difference in surface texture over the three tests. The softer the clay the smoother the extrusion. The harder the clay the more fractured or cracked the extrusion surface appears. Significantly it is these fractures that open up more if the form becomes stressed or distorts. Second is the small but noticeable difference in size due to shrinkage of the three tests once bone dry. The greater water content of the soft clay resulting in more shrinkage.



Slight variation of surface texture where the softer the extrusion the smoother the result.



Bone dry variation in shrinkage indicating difference of water content.

Edge, Texture, Overhang Test Shape

The more complex sample shape illustrated just how unsuccessful this clay recipe was for extrusion printing. However it was not totally unsuccessful in that it offered the opportunity to test what improvements could be made to improve the clay mix.

The tests included a three layer base that printed successfully in each consistency of clay. When printing vertically, as with the cylinder tests all three clay samples built up the form as expected. As can be seen from the photographic documentation with this shape no sample printed clean. The soft clay became unstable with the nozzle movement over the repeat texture area and then totally collapsed as it could not hold its own weight, particularly on the indentation overhang. The soft extrusion did not break, it just did not offer enough structural and directional rigidity as the contour of the shape became more complex.



Two views of the soft, medium and hard clay print tests. Left to right - soft, medium, hard clay. Top image illustrates the repeat relief texture side of the test and bottom image the 45° angle indentations test side.

The medium consistency clay was most successful and printed the repeat texture area but broke down at the extreme of the 45° degree angle indentation. The clay was rigid enough to hold its shape overall but did not have enough plasticity or stretch as the printed wall sagged in the extreme of the overhang. Once the print broke down it never repaired itself, breaking progressively shorter on each layer.

The hard clay seems to have no elasticity when under stress and breaks easily. The print kept building due to the stiffness of the clay extrusion but not layer by layer in the correct place and was less successful than the medium consistency clay.

It would appear plasticity and elasticity are important qualities in a clay to build more complex shapes than the cylinder. While shapes are vertical as in the cylinder tests and the layers are built up with the support of the previous layer underneath, then extrusion printing is alright. But curved shapes that move off the vertical become more of a challenge. Printed infill could be a solution but the object of these tests were to gage what is possible without infill support. Double wall printing is also more stable but again the objective was to test the material qualities. The print process was kept simple to concentrate on what improvements could be obtained from the clay mix. The exploration of printing techniques was beyond the scope of these tests.

Soft clay is more flexible and less prone to surface cracking but then less capable of supporting its own weight. It would appear an improved clay body would have a stronger surface tension, a good

plasticity while also having an inherent self structure to hold its own weight and not fracture under distortion. While printing once a break occurs it is difficult to repair the break and get the structure stable again. The answer is not to get the break in the first place.

Additives to Improve Plasticity and Elasticity

From the unsuccessful print results above it was decided to try and improve the basic test receipt through the introduction of additives. Each additive was tested individually in the original test mix.

Deflocculation

The viscosity of a clay can differ when alkaline, neutral or acid. This is due to electrostatic charged bonds between clay particles. Alkaline additions can result in the dispersal of these bonds resulting in a more fluid clay mix. Sodium Dispex was used in this test. If too much is added it can result in the clay mix taking on a thixotropy character that while desirable for clay slip casting is undesirable for clay printing. A fully deflocculated thixotropic clay does not layer up well when printed and offers no 'bite' to the screw in an auger printhead.

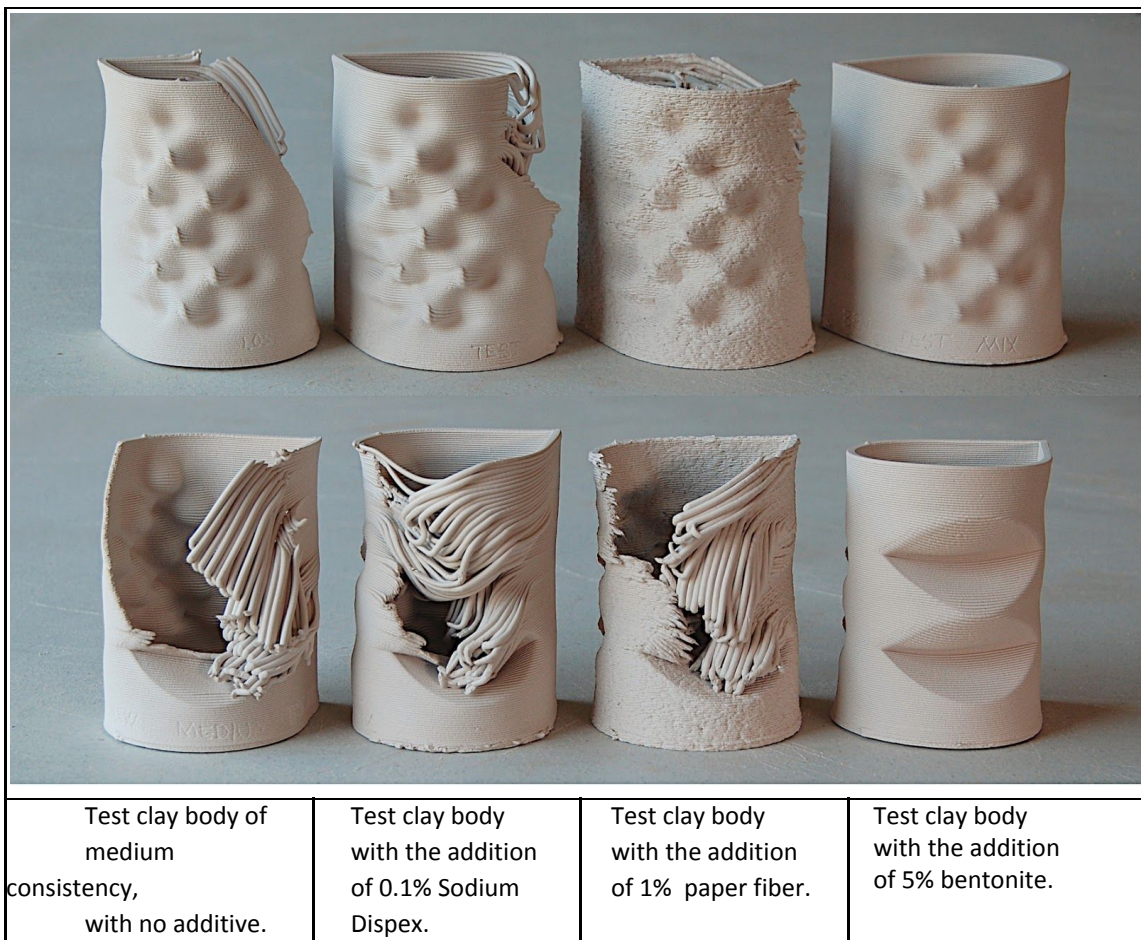
The objective of a small amount of alkaline deflocculant was to improve the extrusion surface quality. The smoothness of the extrusion. An inclusion of 0.1% by weight of Sodium Dispex to dry ingredients weight was decided upon with the aim of slightly reducing the surface tension of the clay mix without greatly changing the character of the mix. This proved to also make the clay mix a little more fluid without the addition of any more water. The resultant test print was an improvement on the initial test but still did not produce an unbroken sample.

Paper Fiber

The inclusion of paper fiber into the clay mix was to see if the fibers would help prevent the extrusion breaking. Concern that excessive fiber might create blockages, 1% by weight of toilet paper to the weight of dry clay was calculated. The paper was first broken down in the required amount of water to produce a consistent consistency of clay. It was surprising how much even this small amount of paper altered the texture of the clay mix. The paper mix printed slightly better than without paper but the fiber texture changes the surface look of the test considerably.

Bentonite Plasticiser

Bentonite is a fine highly plastic very sticky clay. As a final additive to be tested, 5% bentonite by dry weight was added to a sample of the initial clay recipe. Finally a complete print was obtained confirming the importance of plasticity in an extrusion 3D printing clay body, if printing curved or overhanging shapes. It was quite a revelation how the addition of 5% of a material can make such a difference.



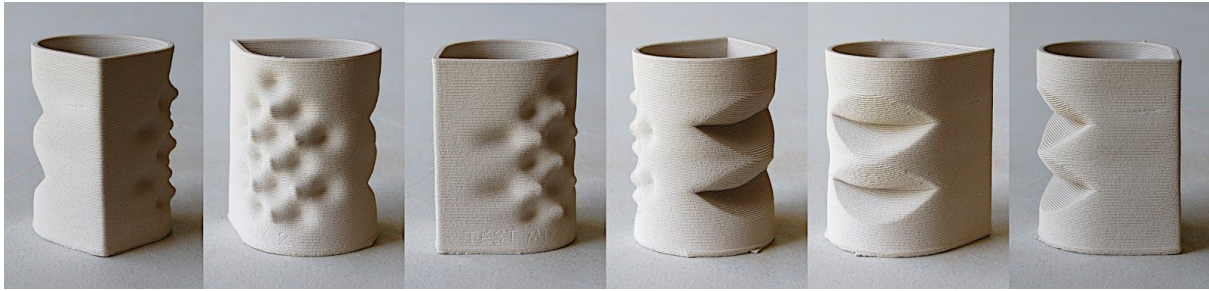
Reformulated Receipt: Test Mix 2

From the additive results and the observation that Test Mix 1 had a high amount by volume of china clay to ball clay in it, a reformulated Test Mix 2 was prepared. For this mix the added paper was reduced to 0.1% of dry weight. The china clay to ball clay ratio was balanced to be equal by volume and the grog, half the volume of any one clay or one quarter of the combined clay volume. This new clay mix was prepared by weight. Water was added in the proportion of 74% dry weight to 26% water, mixed and left overnight.

Test Mix 2

Ball Clay (Hyplas 71)	52
China Clay (Grolleg)	24
Grog (Fine Molochite)	24
Bentonite	5
Sodium Dispex	0.1
Paper	0.1

This new mix produced an adequate print but there are other commercially available clay bodies on the market that produce better results on these test shapes and using this equipment. In fairness to this mix it will improve hugely with ageing and may well prove to be the basis of a successful extrusion printing clay body in time.



Sample printed with Test Mix 2

Test Clay Mix Comparisons

	<u>Test Mix 1</u>	<u>Test Mix 2</u>
Dry ingredient to water	72% : 28%	74% : 26%
Spike Test	32 mm	30 mm
Syringe extrusion	6.3 kg	7.2 kg
Print Pressure	3 Bar	6 Bar

4 - Testing Fine Porcelain Extrusion Printing

Introduction

The aim of these tests was to gain an understanding of how to extrusion print with a 0.6 mm nozzle. It offered the opportunity to test layer height to nozzle size to wall thickness ratios at the smaller end of extrusion range. It was also a chance to try different screw sizes and consider the 'fit' of the screw in the printhead. Not surprising fine printing is very prone to blockages in the clay flow from the printhead nozzle. In these tests I was looking to gauge whether this was primarily a material problem, the material particles blocking the flow, or whether it is a mechanical problem of compaction. Where if too much force is exerted while trying to force the material through a small nozzle the material compacts and blocks.



Clay Samples and Clay Preparation

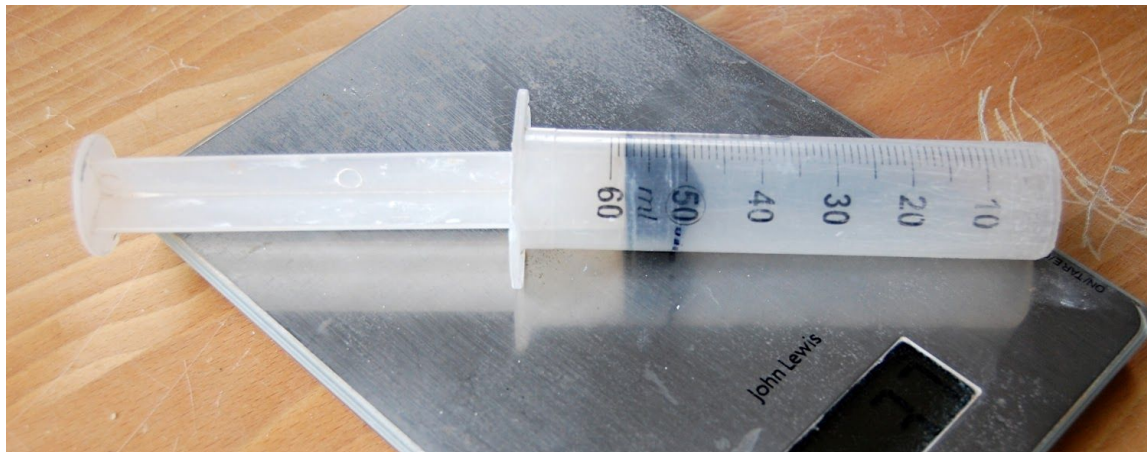
A recycled mix of porcelain clay was used as it was already soft. This I ball milled for about two hours. A ball mill being like a stone polishing tumbler machine. Mine is a one litre barrel container that is filled with the liquide clay and I use glass marbles as balls. This is set to roll on the machine for about an hour. My aim was not to crush or ball mill any particles but just to make sure the mix was thoroughly blended and smooth. This was passed through a UK #120 sieve that I believe has holes/squares of 0.125mm or the equivalent of 125 microns. A porcelain clay is composed of china clay, feldspar and silica. From reference, feldspar and silica used in the ceramic industry are crushed to particle size 30 microns and less. China clay particles are 10 microns and less so a well mixed porcelain liquide slip should and dose easily pass through a #120, 125 micron sieve. Further these materials will easily flow through a 0.6 mm or 600 micron nozzle. This sieved clay was left to dry out to the required consistency taking about two days.

I tested three consistency of clay - that I called Soft, Soft/Medium and Medium. I started testing Soft first to ensure I would get some result, then progressing to Soft/Medium and finally Medium. My so-called medium consistency was similar to what I would use as 'standard' in other tests and for general printing.

Soft	<ul style="list-style-type: none"> • This was a soft paste that I would normally not consider using for printing but it was far from runny. • My 'Drop Spike' stiffness measuring tool dropped straid through the sample and hit the table. So I devised a 'Sinking Spike' measure. The same spike (200 gm) sank 42 mm into the sample when placed on the clay surface rather than being dropped from 10 cm. • I have tried to devise a Specific Gravity measuring system and that calculated to SG 1.72. (See tool below)
Soft/Medium	<ul style="list-style-type: none"> • Again a consistency of clay I would not normally print with but only just on the soft side of what I might consider. • It was stiff enough for the Drop Spike to work and that registered at 45 mm. (35 mm to 30 mm being more normal for my working consistency)

	<ul style="list-style-type: none"> Specific Gravity calculated to SG 1.76.
Medium	<ul style="list-style-type: none"> For this mix I tried to keep to a 'standard' as used in other tests and what I might normally use. Drop Spike 30 mm (As was for the Six Clay Tests) SG 1.78

Specific Gravity Measurement Tool



- I have taken my open ended syringe, stood this vertically on the scale, moved the plunger to contain exactly 50 ml/gm of water and marked it. The empty syringe is 39 gm.
- I can then fill the syringe with my clay sample, push up the plunger to the mark and scrape flat the open end.
- Then if I have this correct, I weigh the clay filled syringe, subtract the weight of the syringe and then divide by the volume of the measure (50 ml)

I do not have a scale that goes lower than a gram so my results I feel are only good to two decimal places. Also my measured sample is rather small so any slight inaccuracies will be enlarged. The result being that I am not sure that the specific gravity measurements are accurate.

Below are two photos to try and help give an impression of what the Medium Mix clay samples looked like that I measured to a SG of 1.78. I then tried to get a consistency more like the measurements you are recording and the sample right by The image on the right is clay of pottery wheel throwing consistency and by my calculation has a SG of 1.84. As can be seen this has little or no stickiness on the hand.



SG 1.78



SG 1.84

Equipment

I used a small Delta type of printer with screw or auger printhead. The clay feed to the printhead was air pressure driven from a 500 ml plastic cartridge suspended next to the printer. The clay paste was fed from cartridge to printhead via a 400 mm long plastic tube with an inside diameter of 6.5 mm. I spent two days printing and in that time only got one total blockage. On closer examination it was a bit of sponge in the nozzle. I guess this got into the mix during mixing and loading of the clay cartridge somehow.

However in my two days I spent hours seeing weird inconsistencies, flow rates dying away and ending in spluttering dots and dashes. Surfaces that looked extra wet, inconsistent layer heights, uncontrollable flow or nothing. Nothing could be interpreted as a blockage, but I am differentiating between a foreign body that causes a blockage and the extrusion stopping because of a fault of the flow system due to gradual compaction of the clay material. The flow system being the balancing of the force required to extrude the clay material through a small nozzle opening. I used the equipment I did as it is very quick and easy to take apart and to look inside to try and get an idea of what was going on.

My reading is that it is a fine balance to get all the forces acting on the flow correct. At this size of nozzle the parameters or leeway gets narrower and it is my interpretation that compaction of the material is what stops the flow. It's a subjective interpretation but brute force is not the answer here. More force just compacts things more and the problems increase. Here we might be considering the Zen of Clay extrusion printing. You must be at one with your process and instrument.

I was surprised how little force/pressure was required to keep things moving. The softer the clay the less pressure was required from the feed but more screw turn was used. I have an observation that as clay gets stiffer it offers more 'bite' to the screw so stiffer clay requires less screw. I have this debate going as to whether the force required to extrude clay through the nozzle, of whatever size, comes from the force behind the clay feed or the screw. Is the feed forcing the clay through, and the screw just helping it along and controlling the stop stop/start? Or is it the screw that is forcing the clay through the nozzle and all the feed is doing is offering up the clay to the screw? My observations are that it is a combination of both. Altering either, feed force or screw rate, seems to affect the extrusion so in my debate it has to be both are important. After these experiments with a

small nozzle, where success or failure is finely balanced as critical points get smaller, I am all the more convinced of the importance of the relationship between the two.

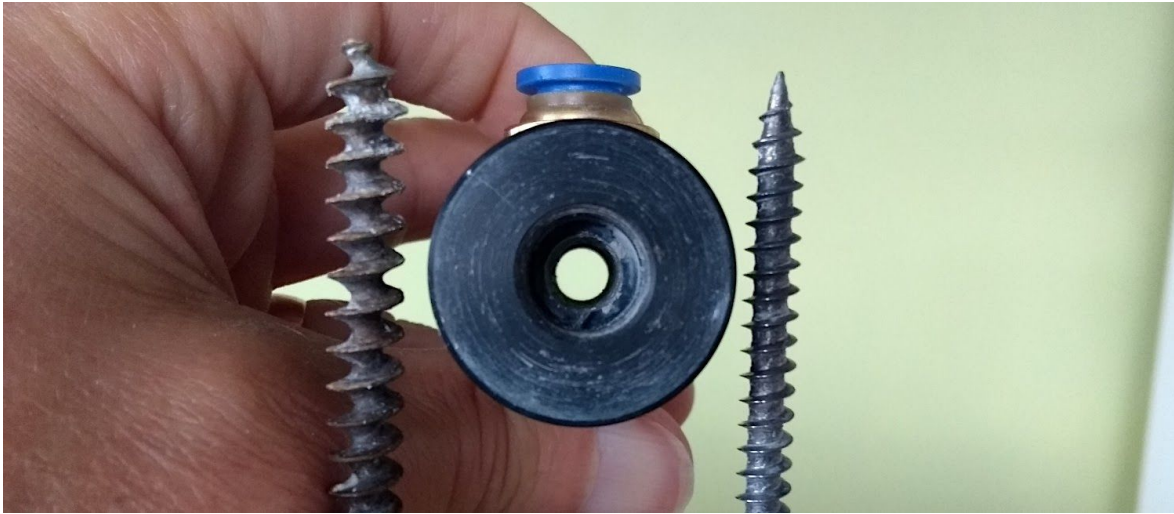
When the extrusion flow was becoming a problem and it was slowing or stopping, I found the compaction I spoke of was at the end of the screw barrel just before the restriction that led into the nozzle. As the machine uses a luerlock nozzle it was easy to detach the nozzle and check for compaction in the nozzle but I felt that was seldom. The problem was where the screw barrel volume restricts into the nozzle outlet.

So the question becomes should the screw go right to this point? Up to now with larger nozzles I have not had a problem in this area but as this was where I was getting compacted clay I took notice. Below is an image of the printhead, luerlock connector and nozzle. (I am holding another luerlock connector so you can see the offending area inside)



Originally the screw only just protruded out of the printhead barrel so when all screwed together there was a gap between screw end and nozzle hole. There was enough screw shaft to extend the screw length and make the screw literally feed straight down the nozzle hole. This has been a rather long winded way of saying that in my mind I have answered the question about whether the screw should go right to the end as it has made a huge difference to a constant free flowing of material. This might have been specific to this model of printhead but it is an important design consideration, for there to be no gap between the end of the screw and any restriction or reduction into the nozzle.

Another question has been whether the screw should be tight in the printhead barrel. Because of the critical nature of working with this size of nozzle it was a good opportunity to explore this. I had two similar printheads and so I could put a screw into one that fitted tight in the printhead barrel (left in the photo). In the second printhead I used a narrower screw that is loose in the printhead barrel (right in photo).



While I had the Soft clay loaded up I did not experiment with changing screws. With the Soft/Medium clay loaded I made three gcode files, all the same shape, the oval (2 x 3 x 2 cm) and sliced as illustrated at 0.4, 0.3 and 0.2 mm layer height.

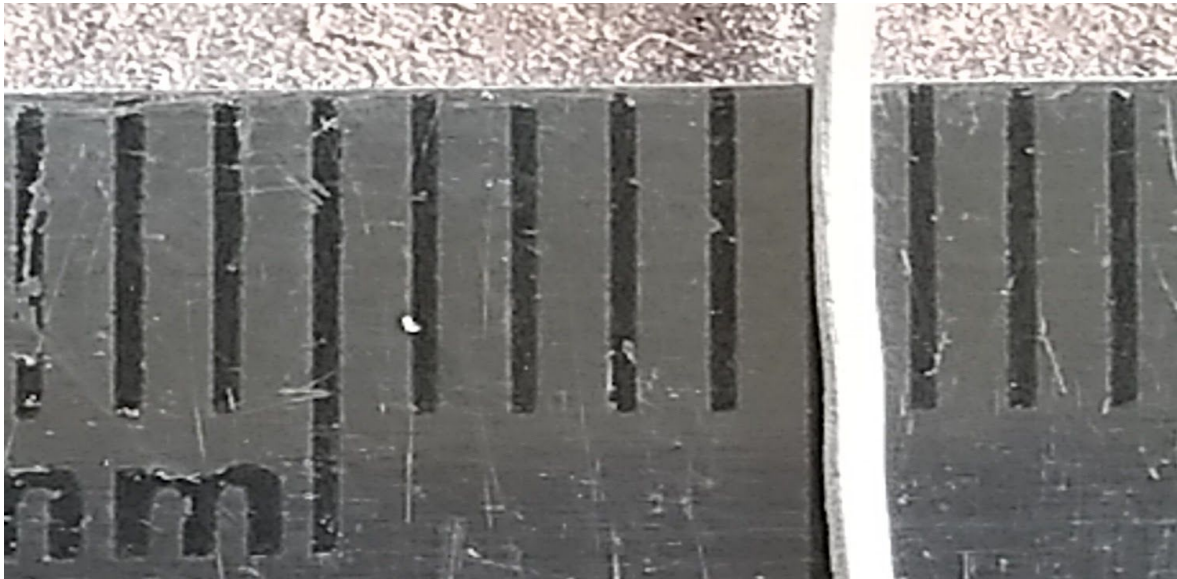


I then printed each file, without making any setting changes using both screws. In the photograph I cannot remember which row, front or back was with which screw but as can be seen there was no obvious difference in the printed outcome whether the screw was tight in the printhead chamber or loose. Considering the volume output of the two screws, the larger screw having a deeper thread, I would have expected there to have been some difference but there was not. So with Soft/Medium consistency clay I had a result and the conclusion was that the character of the screw does not have too great an effect on the extrusion flow.

I then changed to the Medium Clay, still with the wider, tight fitting screw in place. This time there was a much more clear cut outcome. With the stiffer clay I could not slow the system enough to prevent compaction in the transition from screw chamber to nozzle outlet. I was reducing the 'flow' in both the slicing software (Cura) and on the machine LCD control panel and each time after cleaning, the flow would gradually fall away over the duration of a print because of clogging and compaction. My reading is that this is because of too much pressure or force being created inside the printhead by the tight fitting screw. I changed to the thinner screw that is loose in the chamber and it was this combination that proved to be successful for the remainder of my tests. With the

stiffer clay the loose screw was not as powerful as the tight screw. My explanation being the point I made about 'bite'. As the clay gets stiffer the screw becomes more efficient at pushing through the clay.

Process



Clay extrusion from 0.6 mm nozzle

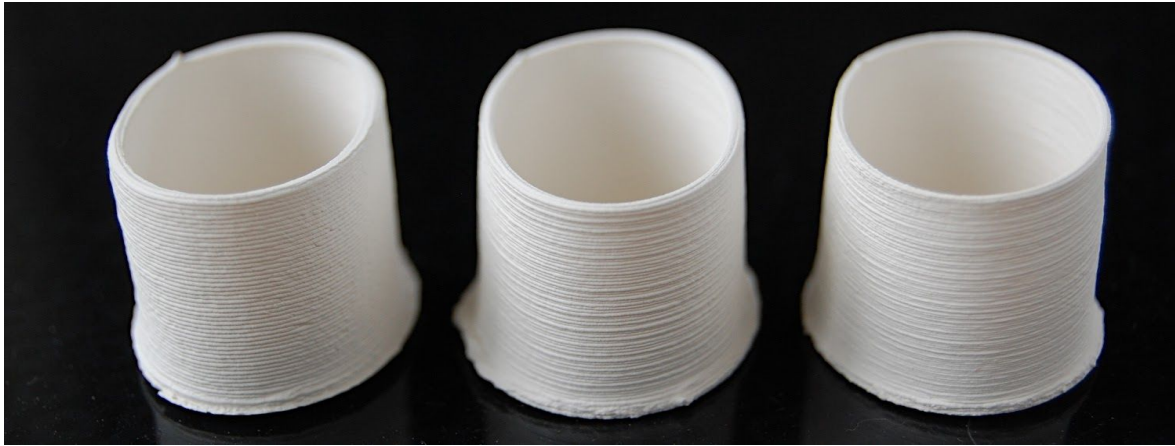
I could not get the **Soft** clay to print well. (Photo below) I started with this clay to make sure I would get some sort of extrusion. The problem I had was controlling the flow. With more fiddling I might get a better result but I was sure I could get better results from the Soft/Medium clay so moved onto that. With the Soft clay there was a lot of distortion of the form and inconsistent flow.



Soft clay print tests

At first with the **Soft/Medium** mix I was getting extrusion stopping because of printhead compaction. It appeared that I needed less force than I expected to get clay through the system. After cleaning out it would flow but gradually things would clog up. Once compaction starts the blockage just progressively gets worse. My solution was to turn everything down and slowly increase the clay delivery with the air pressure and an appropriate amount of screw. Over screwing was a

common problem. I was surprised how little air pressure I required. I was working at 2.2 Bar that is much less than I normally use to deliver clay. Normal delivery is more like 4 to 6 bars of pressure. Porcelain does take less pressure but I did not expect it to be that low. I would expect little or no dewatering in the clay container at this pressure. (At pressures above 4 bar the water begins to separate or be forced out of the clay mix) For the Soft/Medium clay the flow rate on the screw was at 40%.



Soft/Medium mix - layer height test: left 0.4 mm, middle 0.3mm, right 0.2 mm

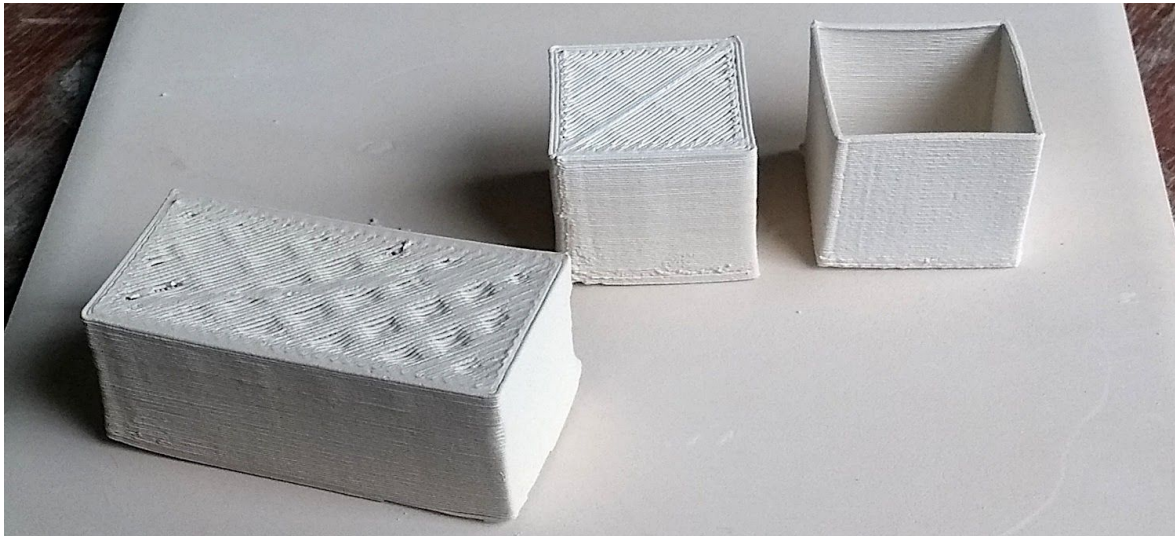
With the system working I began some layer height tests. I started with a 1 height to 3 width ratio that is the ratio I use with a 2 mm nozzle. So with a 0.6 nozzle the layer height was 0.2 mm. (Right hand in photo) Printing with a single skin to test for the stability of the wall this printed alright but I felt it was over compacted. My next test was with a 0.3mm layer height representing a 1:2 proportion. (Middle in photo) Interested to see what a 0.4 layer height would look like I did this too (Left in photo). To the naked eye there is a difference but as this is not significantly noticeable between the three prints I have continued with the 0.4 layer height as this represents the least print time. The 0.4 mm layer height was also the most stable. In the oblong boxes below the left hand two are printed with a layer height of 0.4 mm and the right hand form with 0.35, an almost unnoticeable difference.



Soft/Medium print tests - over screwing on left hand print. Centre print, layer height 0.4, right print 0.35 mm

Moving onto the stiffer **Medium** consistency clay, further adjustments were needed to get a consistent flow. Once again at first I got blocking as I thought I would need more force for the stiffer clay. The final setting, that for all I know might change if I was to try and duplicate it on another day, settled on a clay delivery pressure of 2.5 bar of pressure. Slightly up on the 2.2 bar used for the

softer clay. This is expected as being stiffer clay one would expect more force is needed to deliver the clay down the narrow feed pipe to the printhead. However the flow rate or screw rate needed to be dropped from 40% as used for the soft/medium clay to 20% for the stiffer medium clay. The screw rotation had to be slowed, otherwise over rotating patterns result on the print surface and at worst compacting happened in the printhead. Certainly when printing the 'Top' layer over the infill a further reduced flow/screw rate would be good to avoid slumping of the extrusion.



Medium clay print tests

Conclusion

So the clay preparation procedure of ball mill mixing and sieving.

I like the idea of a tight screw but on this evidence one would need different sized printheads for different sized work and nozzles.

When working at a larger increasing nozzle size my preference is for the ratio of layer height to nozzle size to increase. Here it was decreasing and quite considerably.

While the system remains in balance and there is no compacting of material anywhere things flow but as soon as problems occur they just get worse.

5 - JK Drop Spike - Clay Consistency Measuring Tool

This simple dropping spike tool has proved to be surprisingly consistent and accurate for measuring the consistency of clay paste. The principal is to drop an iron spike onto a clay sample, down a guide tube from a constant height. The length of penetration into the clay averaged over a number of attempts is recorded and gives a measurement read in millimeters. This 'Spike' measurement offers an analytical tool for comparing different clay mixes or offering a measurement technique for keeping constant consistency between mixes. So that measurement can be compared between users the tool has been standardised below.



The spike is made from a length of 10 mm diameter iron reinforcing rod. The spike end is ground to a 60 degree angle. Once the end is shaped and smoothed, the length of the rod is determined not by length but by weight of 200grams. (As a guide my spike is 328 millimeters long) Finally a drop guide line is marked on the spike 100 millimeters from the tip. A drop guide pipe is then made which has a length of 200 millimeters. My guide pipe has an inside diameter of 13 mm and is made from a copper gas pipe offcut. The material and diameter of this guide is not important as long as the spike drops unhindered inside.

To use the tool the 200 mm guide pipe is placed on the clay sample or mix. Place the spike inside the guide pipe so the spike guide mark is level with the top of the guide pipe. The spike will then be 100 mm above the clay inside the pipe. Drop the spike, ensuring the spike sinks no further into the clay, lift the guide pipe and then with a turn withdraw the spike from the clay. The turn helps make a definite clay mark on the spike. Measure from spike tip to end of clay mark to see how far the spike has sunk into the clay.