A Guide to Clay 3D Printing

By Jonathan Keep (Jan 2020)

Clay 3D Printing

This outlines the features to be considered when setting up a clay 3D printer.

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 stuff and the machine techie stuff, 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.

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.

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. If you want to produce lots of the same thing then traditional ways of working like moulding are 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 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.

On the whole 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 it is not uncommon for the printbed to move during printing, so be assured that your style of work will suit this action. Also that if you hope to print complicated shapes at quick speeds that the movement can cope with this and does not judder itself out of alignment.
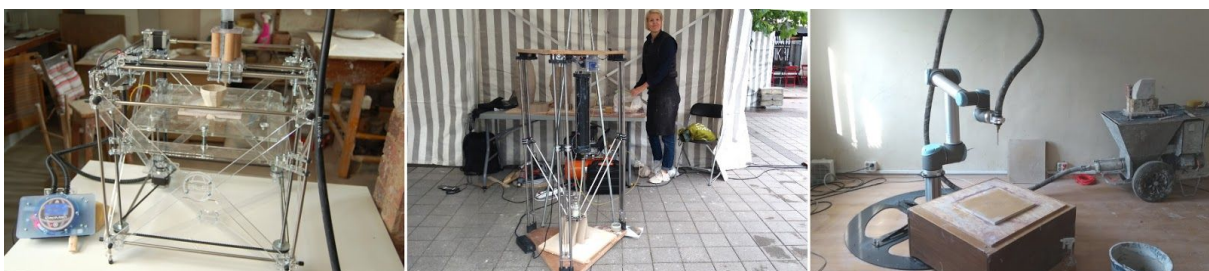
Printhead

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.
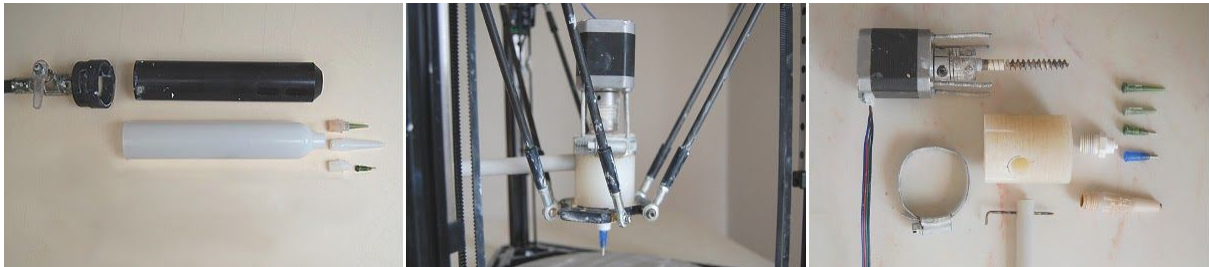

Delivery

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 dewaters 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.
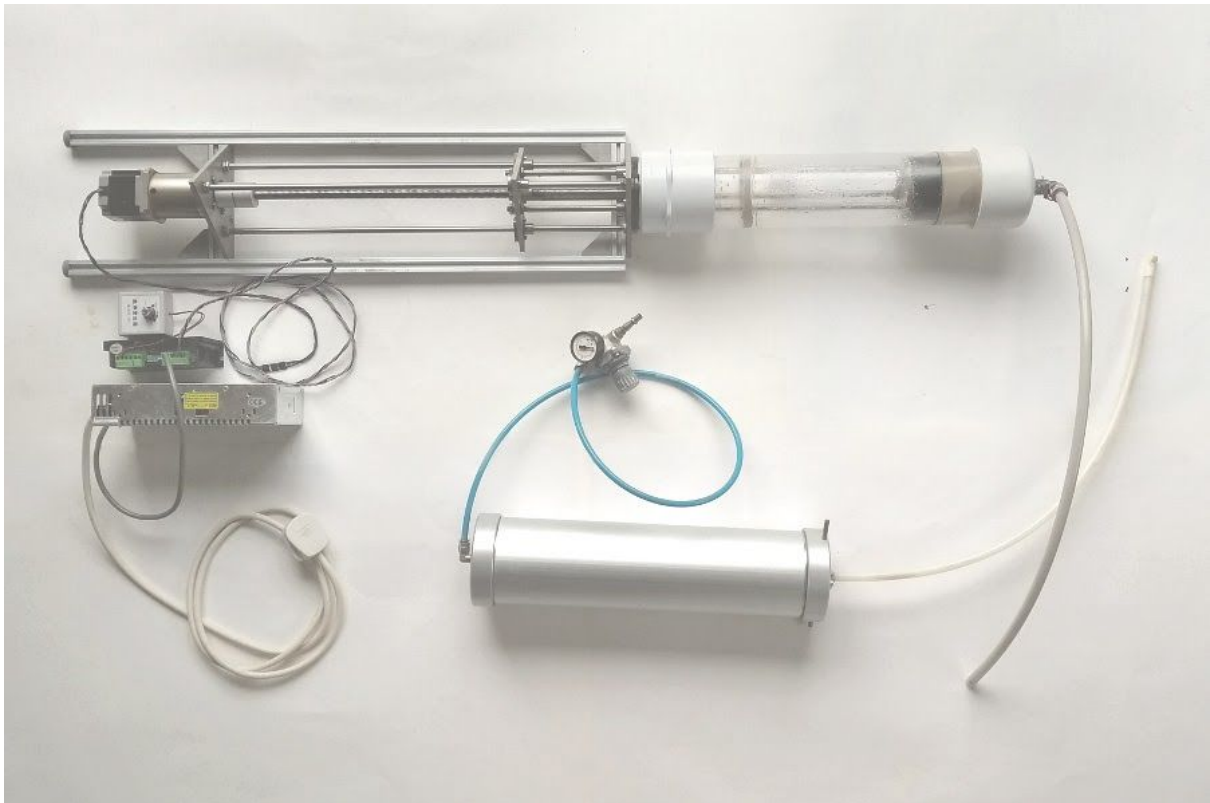
Slice Layer

The slice 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. You might prefer a more rounded look to the print layers but 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. Then 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 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 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.

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 utill 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. Under the Prepare option 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 with a damp sponge just before printing to ensure that the clay extrusion initially sticks to the board.

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 I need a 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 is trimmed 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. 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 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.

3D Files for Clay Printing

This is a guide to preparing digital files for Clay 3D Printing.
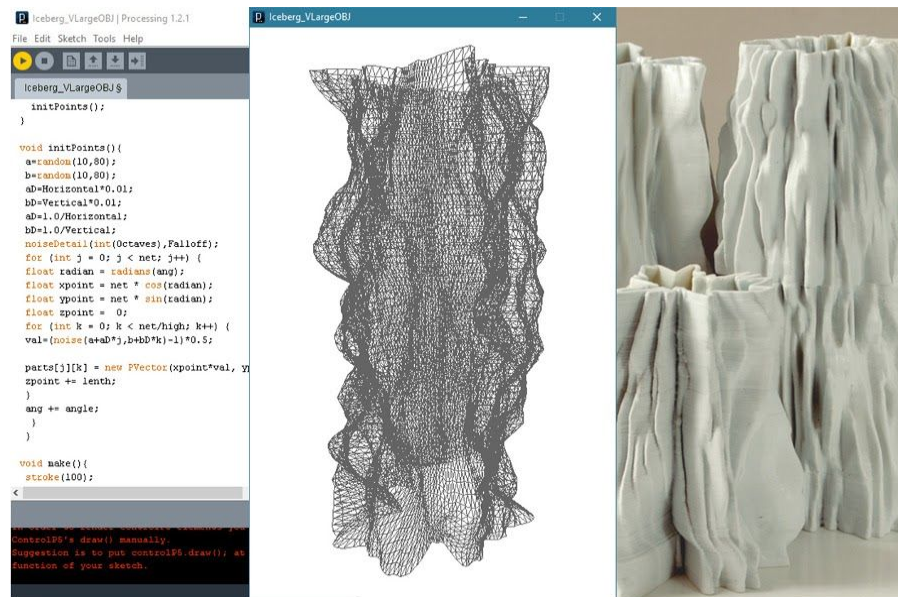
It is going to be your 3D files that you prepare for ceramic printing that will make you stand out from the crowd. Anybody in the crowd will probably be able to work a clay printer but it will be your creativity and individuality that will make what you print special. Well made files print better and 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 decemation 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.
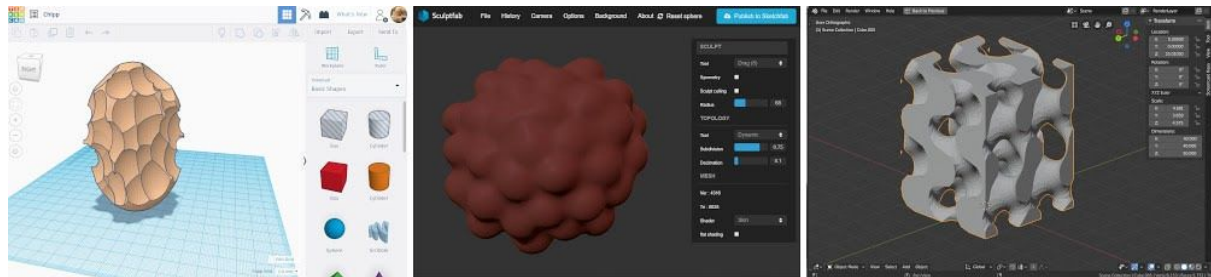
Scanning

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.
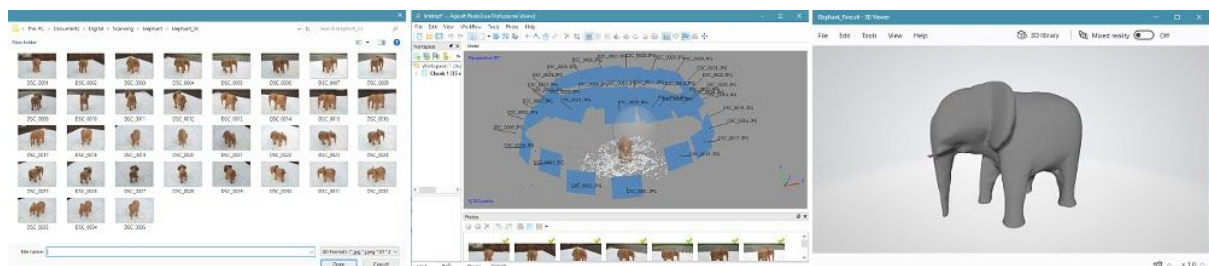
File Download

Copy, cut and paste, or the mash up, offers possibilities that reflect 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,** or **.gcode**, that is the code language that 3D printers understand is 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.
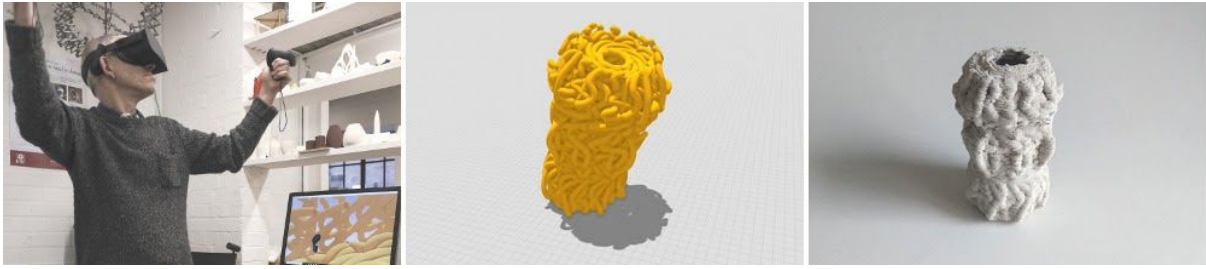
Hints and Tips on Preparing 3D Files

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

A well drawn file produces a good print. Think about the *path* that the printhead will follow and the more continues 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 these problems 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

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 or alter the model. 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 this can be cut away when the print is finished and the clay gets stiff.

File Types

Like the *.jpg* has become the standard for 2D photos, the **.stl** file type has become the standard for 3D printing. More than likely you will be looking to *export* a .stl file from whatever program has been 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 and 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 a 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

An overview of clay for 3D Printing with a guide to clay preparation and clay recommendations.

All clay can be used for 3D printing but some clays print better than others. What you are looking for is a clay that is not too sticky, that has a fine texture and dries quickly. 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 in process from traditional ways of working with clay. In the printer soft and malleable clay is given shaped by building up layers of clay, 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 becomes very messy. Fine red clays are also slow drying. Sandy red clay or red clay blended with other clays will work better.

Cream clays with a good amount of 0.2mm grog/chamotte (15 - 30%) seem to work well. (Grog/chamote is an already fired and crushed ceramic powder/grit) The grog makes the clay less sticky and the mix moves through the system better. It gives a bit more structure to

the mix to help printed forms standing up. Gorged clays also dries 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.

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. If the clay has become thixotropic, when agitated it becomes more liquid then it will not work in a screw printhead and 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 and 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, as for keeping amphibians warm or warm air blowers to help dry work while printing.

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 there is no give between layers then the clay is probably too hard. As I said, you want a good healthy cowpat quality.

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.

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, so chose 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.

Softening a bag of throwing/modeling clay by hand

Take a potters 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.

Mixing from powder in bulk I use a builders double shafted mixer, 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 to the required consistency and to give the mix time to 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 know 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.
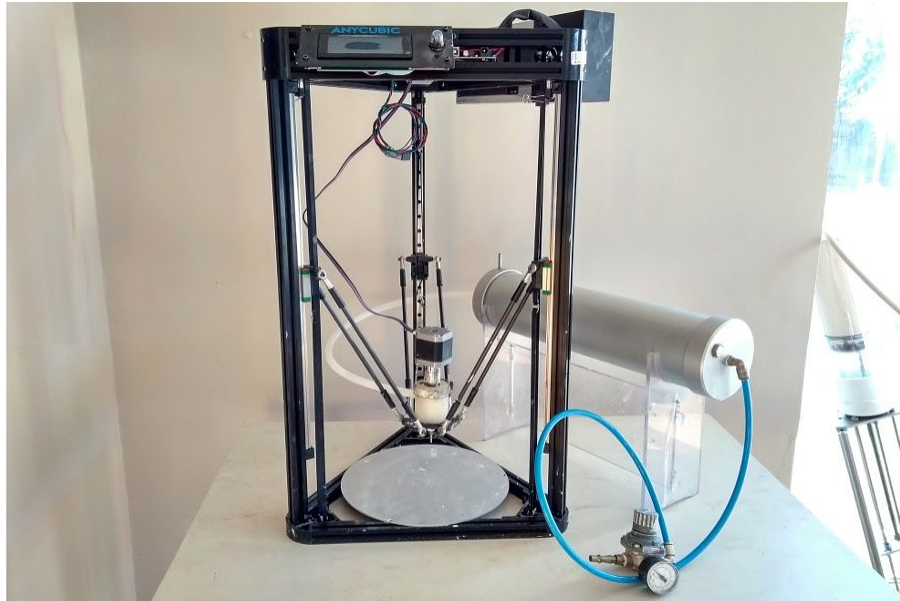
JK Print Mix

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).

Convert an AnyCubic Kossel for Clay 3D Printing

This is a guide to converting an AnyCubic Kossel plastic printer to be able to 3D print in Clay.

In 2013 I published the design of the JK Delta Clay printer. At the time there were not the cheap table top 3D printers available like there are now. Now, the AnyCubic Kossel from China is cheaper to buy as a kit than the cost of 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.

In 2017 Dries Verbruggen documented converting a similar AnyCubic 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

I like to break up the think of clay 3D printers into rig, printhead and clay delivery. By the rig I mean the computer controlled machine that will move the printhead and for this project we use the Anycubic. 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.

Note: AnyCubic maker a bigger delta 3D Printer called a Predator with a build volume of 37 cm diameter 45 cm height, also for plastic. This looks like an interesting option for conversion to clay but I have no further information.

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.
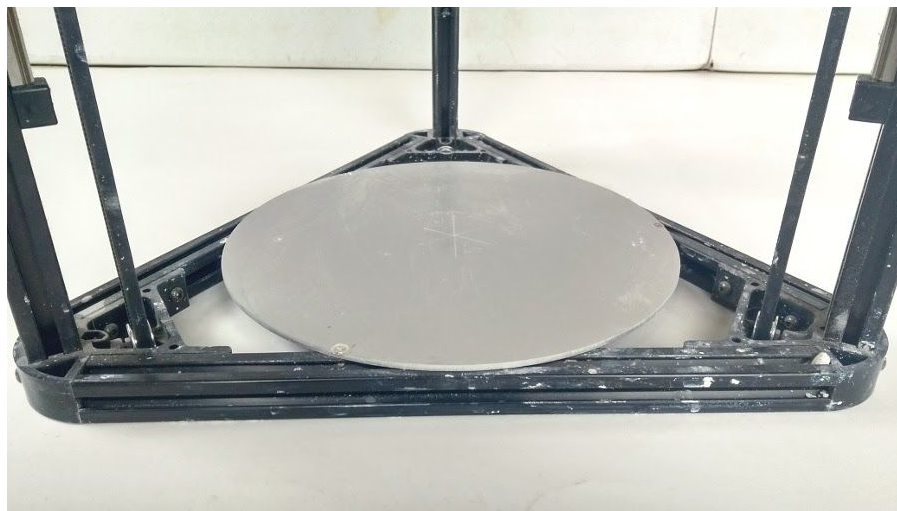
One thing to take note of when you put the pre-assembled bottom frame at the top is that the distance the vertical beams slide into the top and bottom frame. When attaching the Linear slider bars to the vertical aluminum frame bars the positioning gap must now be greater at the top. Space must be allowed also for The Top End Stop bracket. This makes sense when you have the parts. Once you have all the electricks 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.
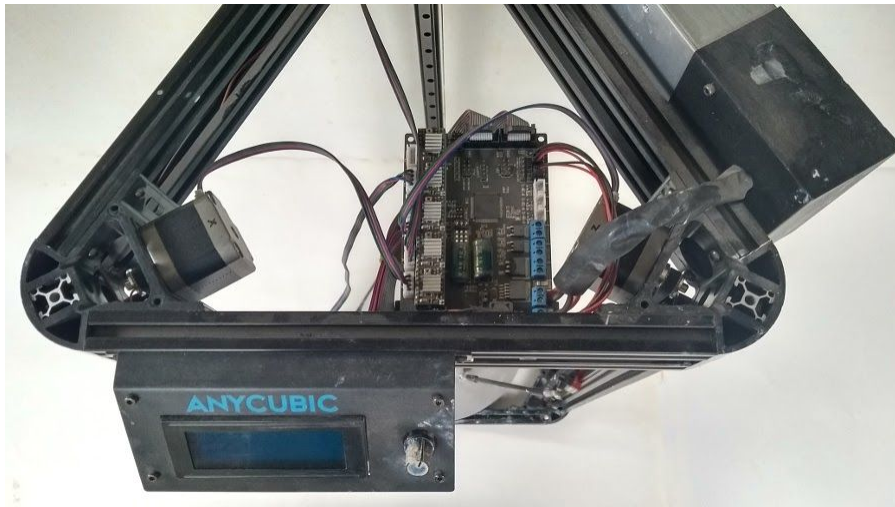
I did move the mainboard along so I could access the usb input next to the display screen. The display screen is mounted left of centre.
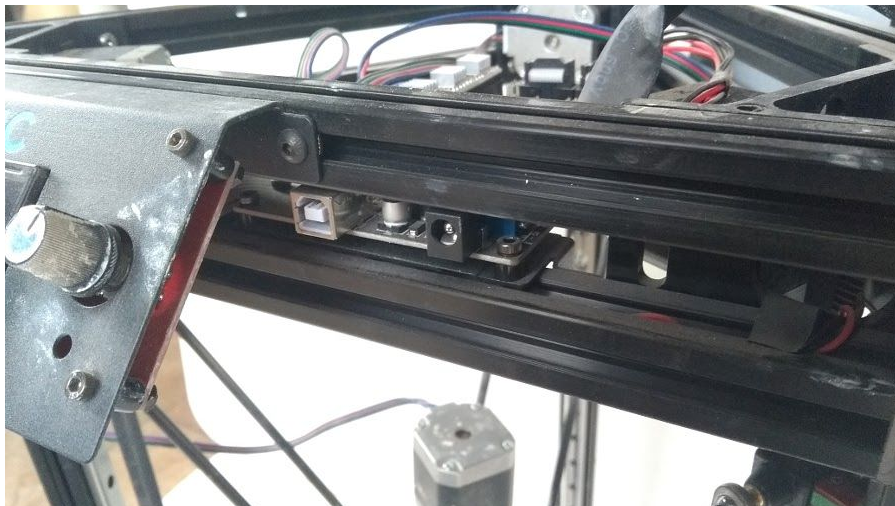
The Power supply I mounted horizontally and up away from any water. The wires just reach the mainboard.

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 and here - https://wikifactory.com/@jonathankeep/self-build-clay-printhead
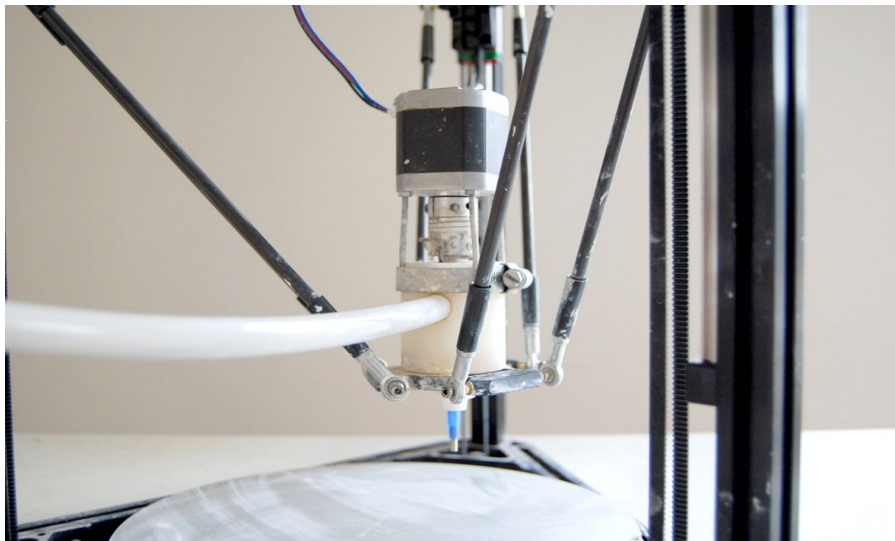
You could also look at -
Stoneflower Clay Printhead Kits - https://www.stoneflower3d.com/
Stoneflower Open Source page - https://www.stoneflower3d.com/opensource/
WASP Clay Printhead Kits - https://www.personalfab.it/en/shop/clay-extruder-kit-2-0/

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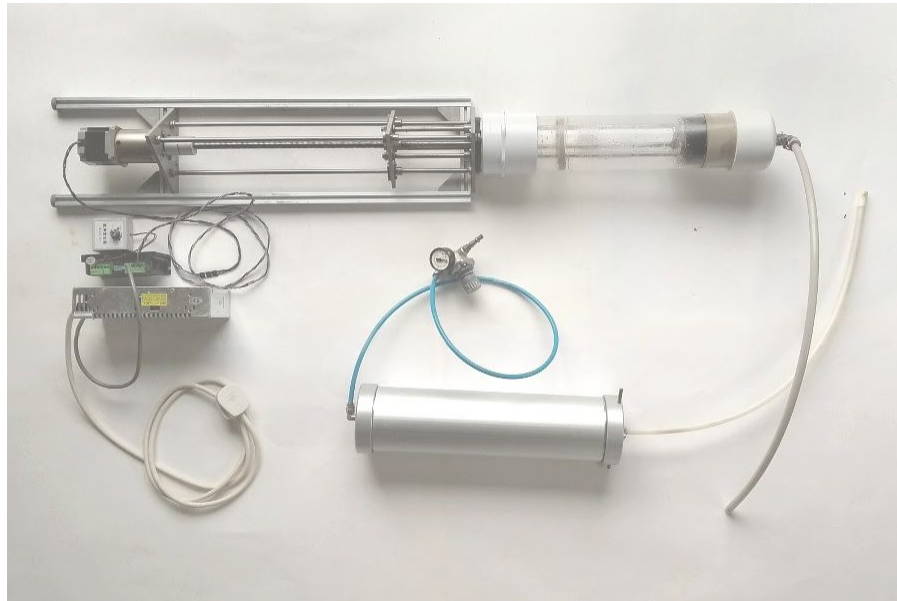
I do also have a mechanical ram per kind favour of ClayBot so I have run my Anycubic with both a ram clay delivery and the compressed air delivery. Both work well but I prefer the simplicity of the air system. I would use the same consistency of clay in either system. In the photograph above the air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

The aluminium container is from the Italian company WASP. These containers are called Bolie guns and are used by Carp fishermen to make ground bait. I also have a similar container that I got in the UK and is made of black nylon and plastic. I use the UK container on a larger printer so it is longer. The aluminium container is a good size for a table top printer and has an inside diameter of 10 cm and is 40 cm long. The endcaps of the metal containers screw on and can become difficult to undo. You need to make sure the threads are clean of clay. The design of the black container that bolts up is easier to use as long as you can get a good seal on the air endcap.


An improvement that I would suggest should be made on the black container design is to have four long rods rather than three rods that hold the two end caps together along the length. The container is put under 4 – 6 Bar of pressure and with only three bracing/connecting rods the end plates bend ever so slightly and the air pressure leaks out. With four bracing rods placed around the circumference of the endplates this could be 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 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 that I am told will release pressure above 8 bar of pressure. A very important safety feature.
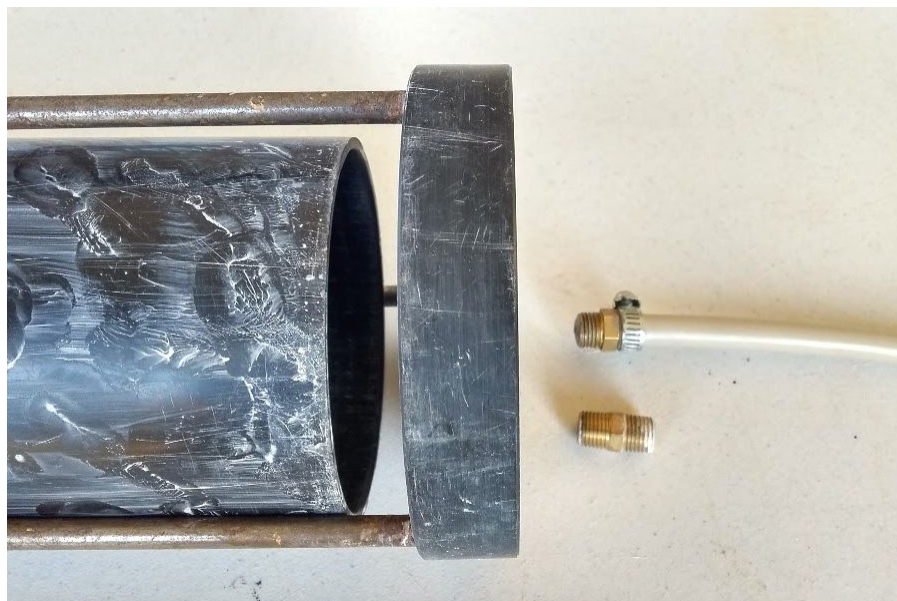
At the clay end there is a nylon pipe that carries the clay out of the container and to the printhead. I use Nylon Pneumatic Air Line Tubing, clear/natural with an outside dimension of 12 mm and inside dimension of 10 mm. This also comes with a 9 mm inside diameter and this pipe is then too stiff. The 10 mm inside is more flexible, that is good but also kinks occasionally and needs replacing. The length of this 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 boilier gun.

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 strait 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 put a flat plate on the container end plate and attach the 12 mm nylon pipe

Self Build Clay Printhead

A guide to making an auger, screw type stop start or retraction printhead for clay 3D printing.
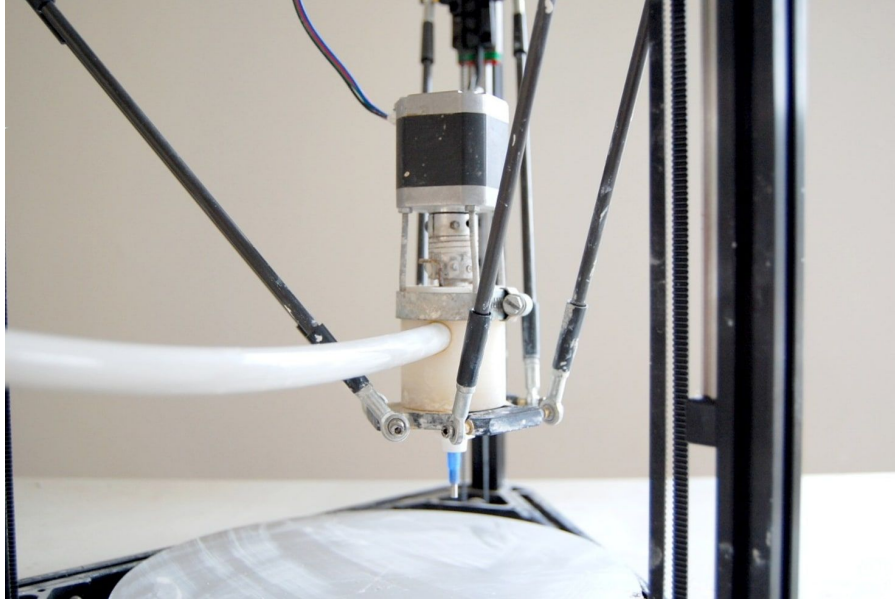
The overriding design principal with this clay printhead was that it would be easy to take apart, to clean and maintain. I do not like plastic printed parts as I have found them not to be strong enough and why go to that length if the parts can be kept this simple. My judgement for the design decisions 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 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.
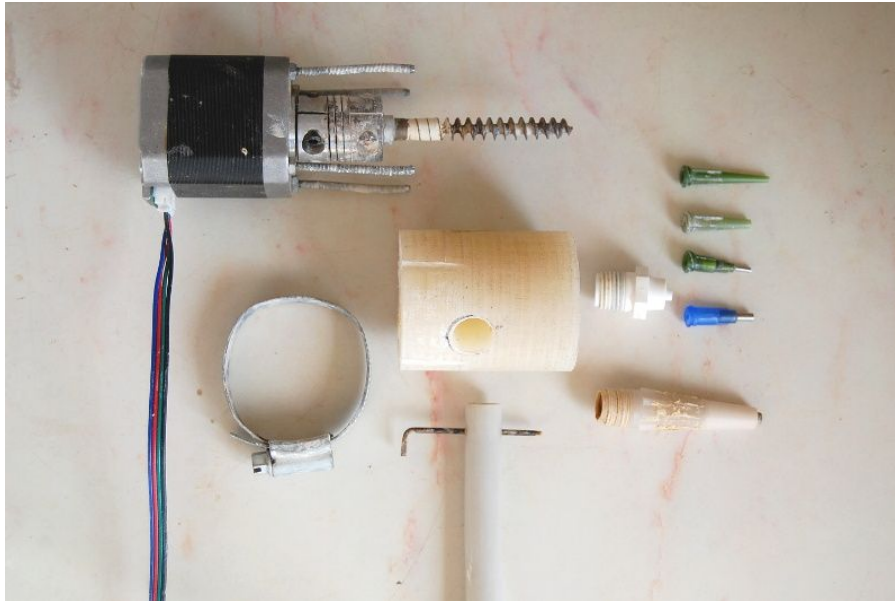
Although the turn of the screw will be driving clay down I feel it is favourable to have the clay feed/delivery under more pressure than the screw will naturally take to help force the clay extrusion. To stop clay rising up the screw shaft I seal the upper thread with hardening putty that forms a tight 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.
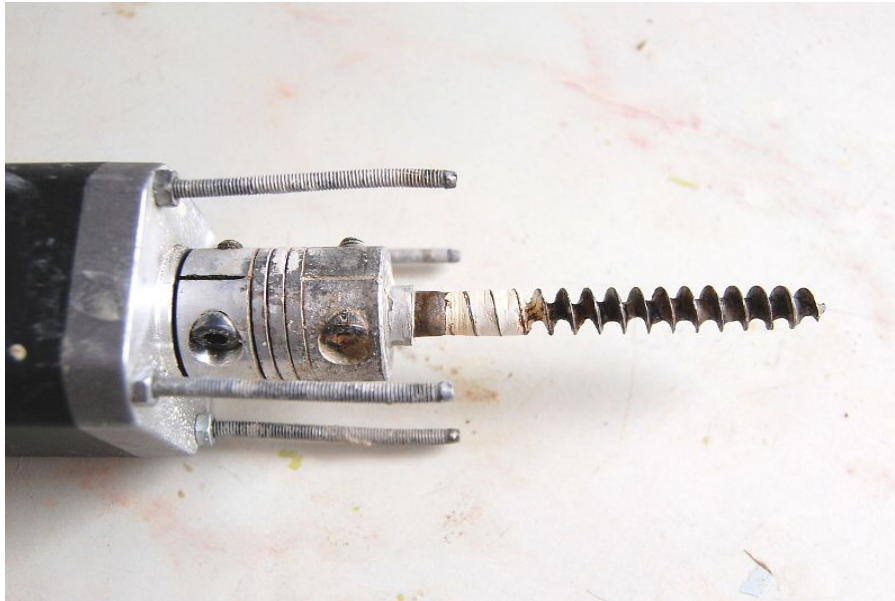

Feed pipe

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. (OutsideDimesion 12mm, InsideDimesion 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.
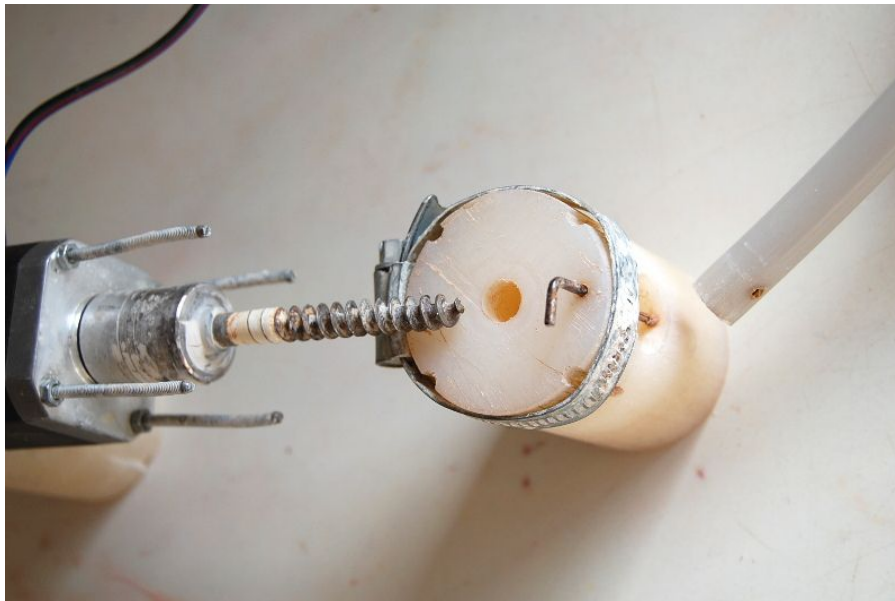
Nozzles

I use a range of nozzle sizes depending on what I am doing so I have kept to a standard ¼ inch threaded outlet hole underneath the printhead. The central hole is drilled bigger and a ¼" 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 ¼" 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
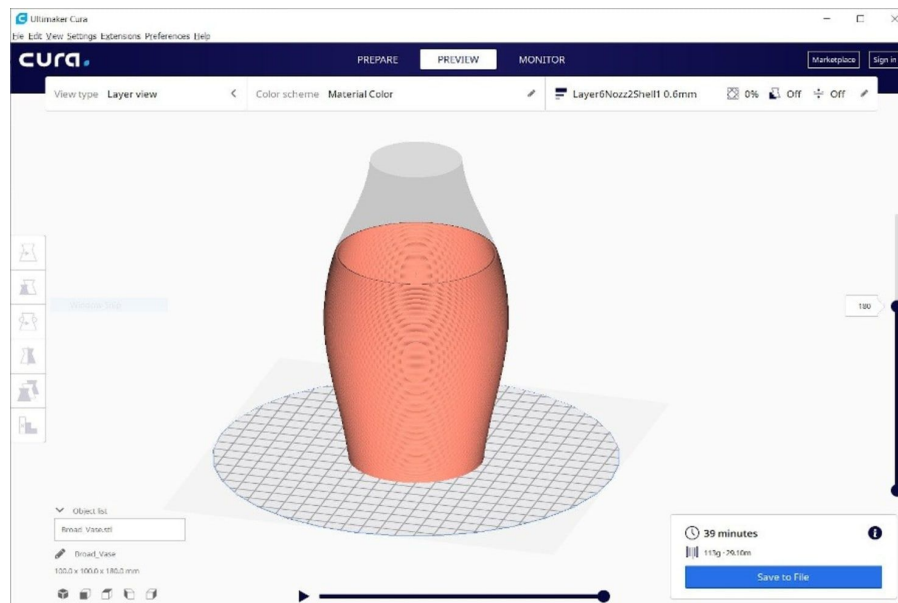
Introduction

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)
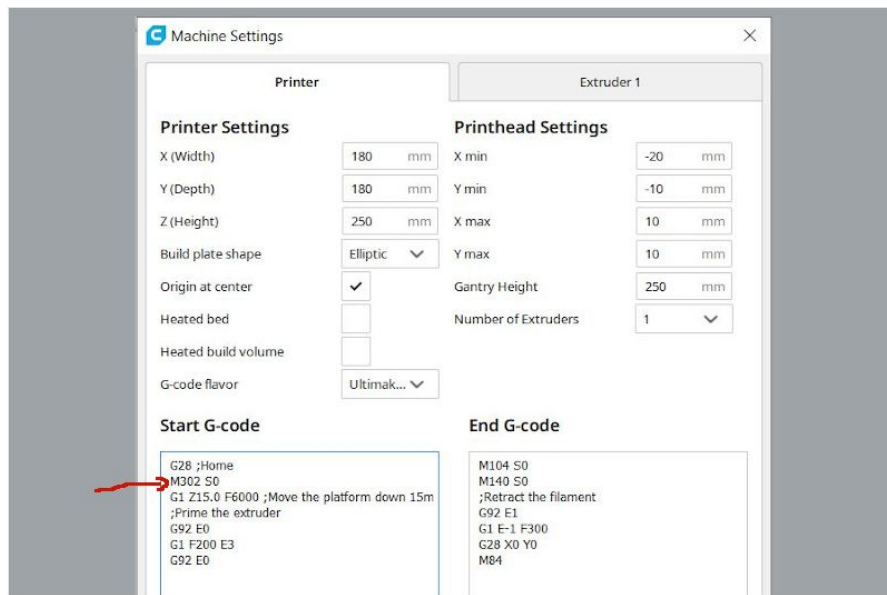
Printhead Settings

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 daufalt 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 Spiralized 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.
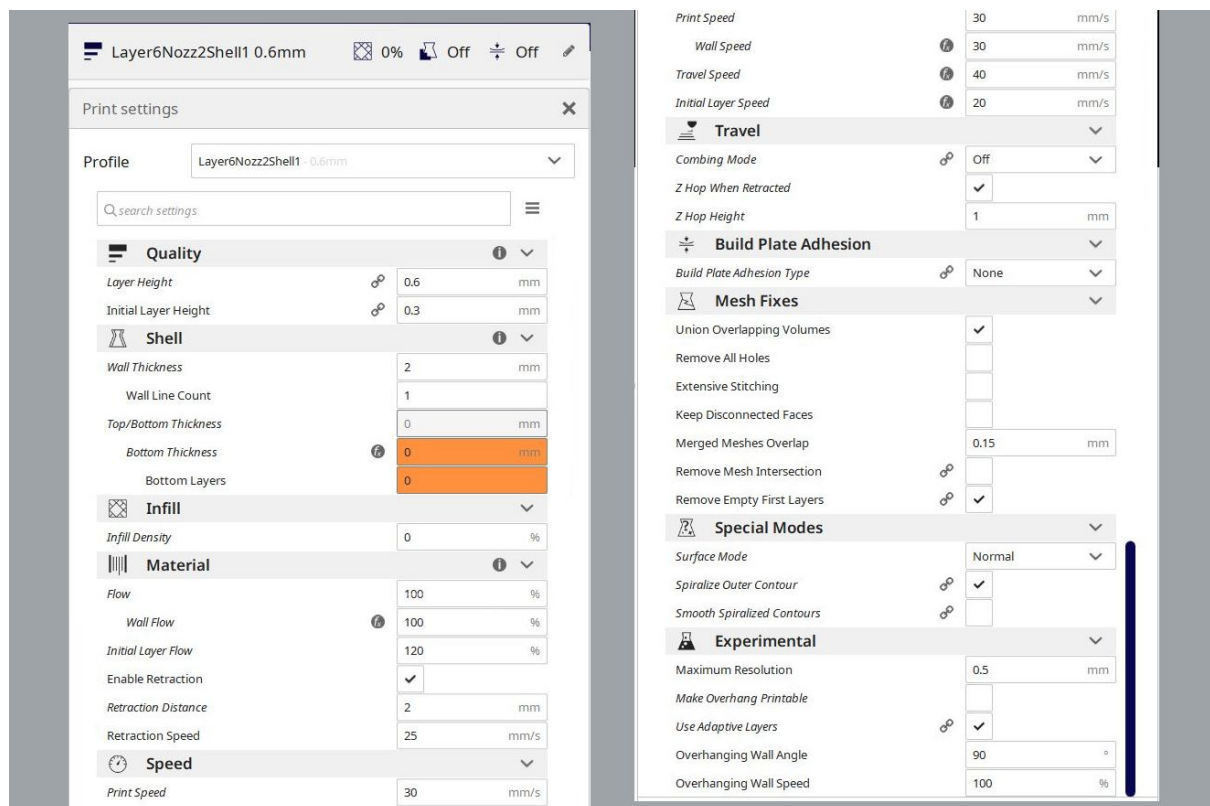
Import Export Print Profiles

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 Work Flow

Once you have setup 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 settings. 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.

Good luck.