Using your Duo

Duo setup

Your duo allows you to print in two colors or two materials. These instructions cover how to set up and calibrate the duo, and how to use it to print in two colors or two materials. Instructions given here are for Slic3r and Repetier Host.

First, in the Printer Slic3r settings under General be sure you have set "number of extruders" to 2.   extruders are set up like this (as seen from the front):

Extruder 1   Extruder 2

Note: duos shipped in February 2014 and later will have these settings reversed- a new instruction set will be posted with appropriate changes after these start shipping.

Then Extruder 2 should have an Extruder Offset. If you were not given the offset with your printer, you can use the procedure here to find it. Extruder 1 offsets should be set to zero.

If they have not been set, or you have lost the values, you will need to measure the offset of the nozzles to prepare for dual extrusion. To do this, make sure your extruders are loaded with two different colors of PLA and run the combined offset test code that can be found at:

http://www.bukobot.com/_media/software:combined_offset_test.gcode

This will print a test pattern to allow you to calibrate the extruder offset. Find the line in the vertical and horizontal test that match up best and look up the corresponding number in the table at the beginning of that file.

In other words, if you are standing in front of the machine, the Y offsets will be from back to front and the X offset, from left to right. If the best match in Y is the farthest back of the tick marks, that would be an offset of -1.0.

The next tick would be -0.9 and so on through positive 1.0 at the end nearest you. On the X offsets, the far left would be an offset of -29.0, the next tick mark is at -29.1, and so on to -31.0. Viewed from above and in front of the machine: (note- the pattern was peeled off a platform and put on a red background for visibility):

Y offsets
-1
-0.9
...
0
...
1
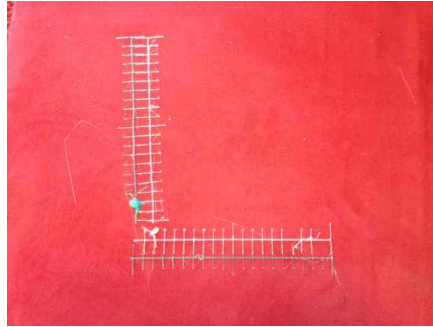
X offsets   -29.0   .....   -30.0......   -31.0

Operational thing that are different when using two extruders

Now you are ready to print with your duo. There are two common cases: using one head to print support material (and the other to print the object) or printing a two-color object. First though some things are different between the single and duo.

- Run the two heads at a lower temperature than you would if you were running the same material at - about 10 to 15 degrees lower than you would for a single extruder running the same material.

- Extruder 1 setting will set the platform temperature so be sure you have those temperatures set correctly on extruder 1 . If you are using different materials think about what the material on the bed will be for the most part. Some experimentation may be required.

- If you are printing in two materials that need different bed temperatures or bed surfaces to stick, somehow arrange for all or most of the first layer to be of one material and use the appropriate bed temperature and material.

Case 1. Using one head for support material.

If you are printing something with significant amounts of support material, you can either use a contrasting color (so you can see all the bits of support) or use dissolvable support. In this case you can use the regular Repetier Host interface to slice your model. You need to change the Slic3r Printer settings to two extruders and set the support material to be one of the extruders.

Case 2. Models that you want to print in two colors.

In this case, you need to do a somewhat different process than you did for one-color printing. You need to create two .stl files that are interleaved into a .AMF file. Note that any rotation needed when arranging the files for printing has to be done in the .stl file generation, since this process does not allow for any rotation in Slic3r.

You also have to arrange things so that there are no places with structure of both colors trying to occupy the space. If something penetrates something else, there must be a hole in the one object to accommodate the second, just as in physical space.

The steps, then, are this:

- First, generate your stl file in two parts, but interleaved as they will be in the print.

    - IN OpenSCAD, for example, if you plan ahead that you are developing a file for two extruders, you can put in a "!" at the front of a structure to include only the structures included in that structure, and a "*" at the front to exclude a particular structure.

    - Example: Code to make two boxes, one poking through the other:

            difference() {
            cube([10,10,2]) ;
            translate([1,1,0])cube([3,3,2]);
            }
            translate([1,1,0])cube([3,3,5]);

        - Change it like this to export the larger square with a hole in it and export that first STL file (note exclamation point ahead of "difference"):

                !difference() {
                    cube([10,10,2]) ;
                    translate([1,1,0])cube([3,3,2]);
                }
                translate([1,1,0])cube([3,3,5]);

        - Change it to look like this (add exclamation point ahead of "translate")to output the smaller box that will penetrate the larger one, and then export the second STL file that will make up this object:

                difference() {
                     cube([10,10,2]) ;
                     translate([1,1,0])cube([3,3,2]);
                }
                !translate([1,1,0])cube([3,3,5]);

- If for instance you had a cup that was going to have its handle printed in blue and its body printed in red, you would export the handle first in one .stl file (hanging in space where it will be on the cup) and the body to another .stl file. Then you would assign the handle file to the extruder with the blue filament and the cup the one with the red filament. Be sure you rotate the entire model before exporting because the combination file does not allow rotation.

- To slice the file, you have to use Slic3r itself, not Repetier Host. Here is the process:
  - Bring up Slic3r, and in the FILE menu select "combine multi-material STL files." Bring in the files, and then CANCEL when you have brought in both files.
  - The filament setting on top is for extruder 1.
  - The program will create a file ending in .amf.xml which combines the two. Set your print, filament (for both extruders) and printer settings. Then click "Export G-Code" to get your G-code.
  - To print your piece, go into Repetier Host (or Octoprint) and import the g-code you just made. Run as normal.
  - If your model needs support, in your Slic3r settings under Support you can decide which extruder's material you want to use for support. This material will be used throughout the model, even to support materials from the other extruder.

- If you are using a duo to print two colors and, on top of that, support is needed, you will need to pick one of the extruders to do the support for both materials.