



DX - Dual Extruder Basic User Guide

12/4/2019

This document assumes general familiarity with using Kisslicer, and 3D printing in general. It is meant to help those already familiar with using the PolyPrinter 229 and 508. It therefore focuses on differences between the 229/508 and the new DX series.

[Basics](#)

[Before you start](#)

[Kisslicer setup](#)

[Printing in a Single Color](#)

[Printing with Two Colors](#)

[Loading the Part](#)

[For parts that also need Support](#)

[Printing with Special Support Material](#)

[Connecting to the Printer](#)

[Removing and Installing the Filament](#)

[Heating the Extruder\(s\)](#)

[Removing Filament](#)

[Installing and Flushing Filament](#)

[Loading Gcode and Printing](#)

[Printing Files that have been previously uploaded to the printer](#)

[Quick Controls](#)

[Maintenance Wizards](#)

Basics

There are three main ways to use the DX printer

1. Single color
2. Dual color
3. With special support material

The main differences between them relate to the complexity and decisions that will be made when choosing the appropriate slicing settings.

Each of the three main choices above are supported by a preset selection in the Printers list in Kisslicer:

1. Polyprinter 465DX Single Color
2. Polyprinter 465DX Dual Color
3. Polyprinter 465DX Dissolvable Support

Each Printer setting remembers the last Material setting, and other settings it was used with, and is therefore an easy way to begin setting up your choices for a specific part.

Before you start

Kisslicer setup

There's a special download required that adds the 465DX-specific setups for Kisslicer.

It is currently a manually-installed ZIP file, to be loaded after the normal Kisslicer package has been installed.

Unzip the contents into the C:\ProgramData\PolyPrinter\KISSlicer folder.

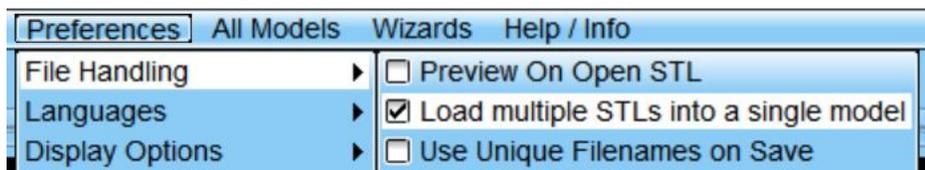
It will overwrite a few files but mostly add. Please let us know, if you get error messages.

You'll need to recheck each of the DX printers setups to make sure the Style and Support and Material selections make sense together for that setup.

Loading Models - configuration

You must:

1. Ensure that your Kisslicer has its Registration Code entered. That will enable the dual head printing options.
2. Set the option that will load multiple STL files as one part.



Printing in a Single Color

Choose Polyprinter 465DX Single Color, and slice just like you would with a PolyPrinter 229. Put the desired color in the left-hand spool holder and extruder.

Printing with Two Colors

Choose Polyprinter 465DX Dual Color.

Loading the Part

What can be printed in two colors?

The part must be designed to have multiple sections, either in two or more separate STL files, or multiple sections within a single STL file.

If it is split into multiple STL files, they must all be loaded simultaneously. Use the <ctrl><left click> on each file name to select multiple files, and then press the Open button.

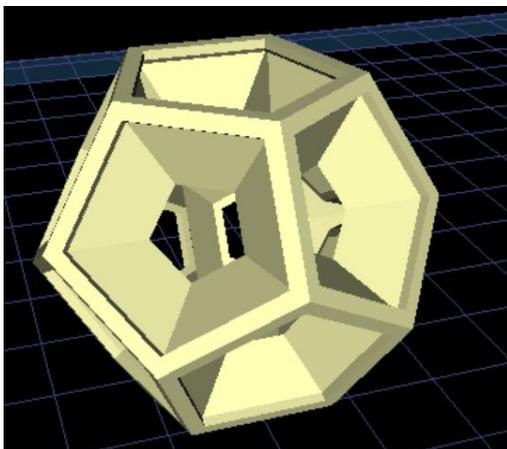
Name

 dodeca-frame.stl

 dodeca-inner.stl

Kisslicer will load all files at once, and co-locate them in the relative positions they were assigned when they were saved.

You should see the part as one whole object, with all the pieces put together:



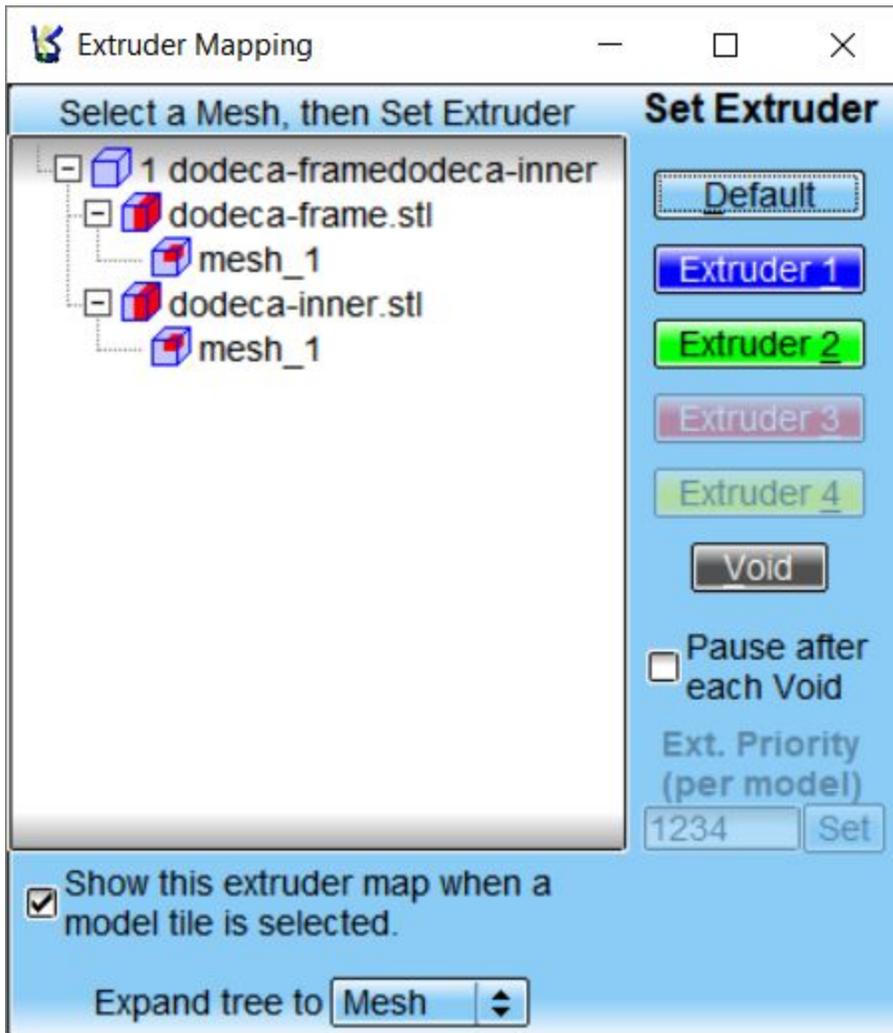
There will only be one thumbnail on the right hand side:



This part has two main pieces - one for each color.

You must now assign each section of the part to one of the two extruders. By default, all sections will be printed by Extruder 1.

If you click on the thumbnail tile, as shown above, you should see a window pop up:

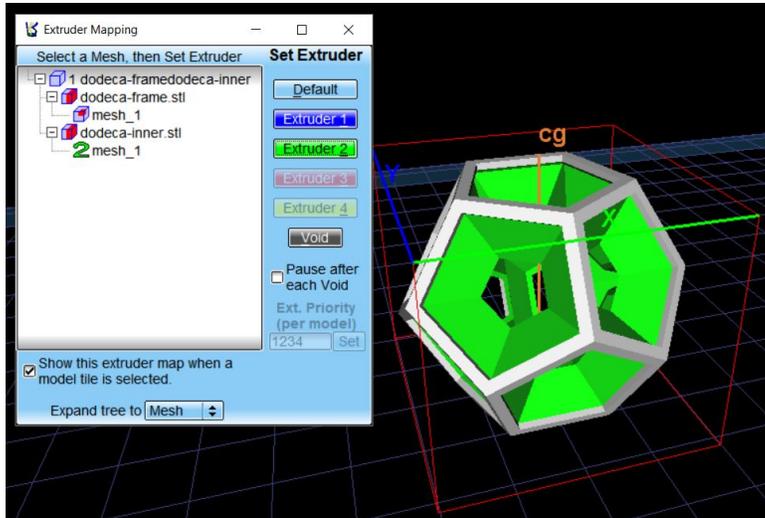


(If it doesn't pop up, then right-click on the thumbnail, and select Show Mesh-to-extruder Map. Then make sure the checkmark, as shown above, for "Show this extruder map when..." is checked. That will make it pop open the next time you click on the thumbnail.)

The example part, the Dodecahedron, only has two STL files, and each file has only one mesh. You can assign an STL or an individual Mesh to a specific extruder by selecting it on the left, and then clicking on either Extruder 1 or Extruder 2.

Since by default, all meshes will be printed with Extruder 1, you really only need to select the one(s) that you want to print in the second color with the right hand extruder, #2.

So we click on one of the STL files and hit the Extruder 2 button:



The sections of the part that will be printed using Extruder 2 are displayed in the main view, and a "2" appears next to the mesh.

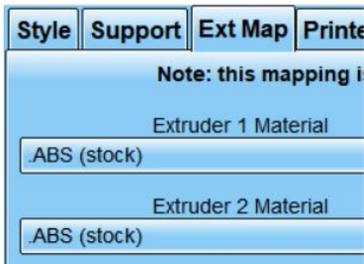
At this point you just need to make sure that the other basics of slicing a part are taken care of. Support

Whenever you are printing with both extruders, it is best to use a Prime tower. The Prime Tower helps make the separate materials very clean and neat. If it's not used, the part may have some extra bits of filament on the outside, or intermingled with the other color undesirably.

The Dodecahedron part prints best with no support, so select the No Support with Prime Tower (stock).

For an initial test print, select the .Normal Style.

Click on the "Ext Map" tab in Settings, and make sure that both of the extruders have the material you intend to print (this should normally be just as it was the last time you were using the "PolyPrinter 465DX DUal Color" printer selection.



You should be able to now Slice and Save the file. See later sections for loading filament and running the print.

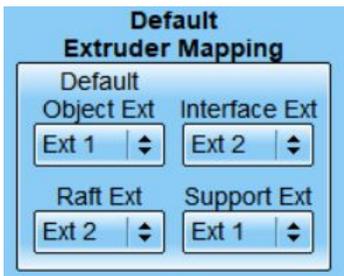
For parts that also need Support

You will need to also select the Extruder to be used with the support.

Normally, one of the two colors will be the majority color of the part in the area that needs support. You will want to select Interface extruder to be the same extruder as the majority part extruder that is in contact with the support, so that you don't get a contrasting support interface color, which can leave small remnants of color and make the part look messier after support removal.

You will want to use the Extruder Mapping section of the Ext Map tab of the settings, and select the desired extruder to be used for the Interface material.

For example, if the part has support, and that support is generated mostly in contact with the color in Extruder 2 (right side), then you'll want the Extruder Mapping to look like this:



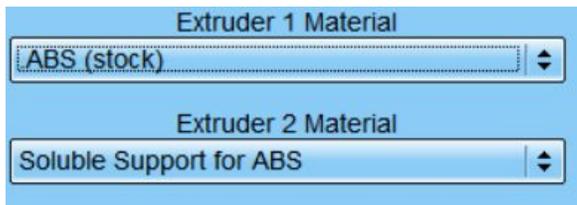
The "Support Ext" can also be set to Ext 2, but if both are the same material, it doesn't matter much, because the main support material generally doesn't come into contact with the part.

Printing with Special Support Material

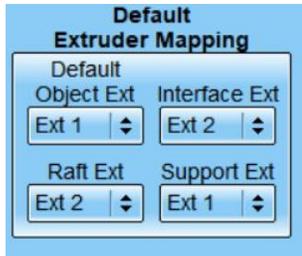
This is usually simpler than printing with two colors, because Kisslicer can fairly automatically generate the correct gcode.

Select the "PolyPrinter 465DX Soluble Support" printer.

You will of course need to know what type of support material you need to use with your chosen main part material. For example:



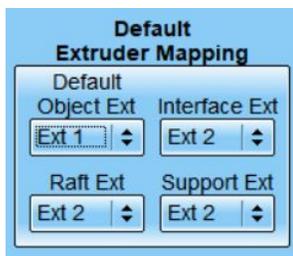
You will most often probably need to only use the soluble support material for the Interface:



But occasionally, if

- There are areas of support where the soluble material doesn't stick well to the main support printed with the main part material in Extruder 1
- Or if the part has internal areas that are filled with support material that will be impossible to extract from the part if not soluble

then you will need to also select Ext 2 as the Support Ext in the map:



Then the entire support structure will be printed in the support material.

Load your part file as normal, and then Slice, and Save.

Connecting to the Printer

Since the normal connection will now be via a network cable, instead of USB, the connection procedure is quite different.

(If the printer has never been connected before, please see the **Getting Started with DX** guide for how to ensure that you have a viable network connection.)

If you have saved the printer as a Favorite in your browser, select it from the Favorites list and it should re-open.

Otherwise, if you know the network I.P. address the printer is supposed to have, enter it into the address bar and hit enter.



Removing and Installing the Filament

If you are familiar with using a PolyPrinter 229 or 508, the actions you will take are going to be very similar, but with some of the things being mirror images left vs right.

Heating the Extruder(s)

You should see, at the top of the web page, the main heater control panel:

Heaters		Control All ▾			
Heater	Setpoint	Set Temp	Set Off	Current / Set	
Left	230	Set	Off	22.2°C / 0	Flush
Right	230	Set	Off	21.9°C / 0	Flush
Bed	110	Set	Off	24.7°C / 0	

To heat an extruder, simply ensure that its Setpoint is above 200 (230 by default), and press the Set button for each extruder you want to remove filament from.

Removing Filament

Give it a bit of time to heat up to 170 or so, and then release the filament by pressing down on the yellow filament lever, while pulling gently upwards on the filament. It should come free.

Installing and Flushing Filament

The left and right filament spool holders are mirror images of each other. The spools mount similarly, but filament comes off of the left roll in a counter-clockwise direction, while coming off the right spool clockwise, just like a PolyPrinter 229/508.

The two extruders are also mirrored, with the left extruder being like a 229/508 extruder and the right extruder being a mirror image.

The filament switches are both like a 229/508.

You will need to pull the extruder wiring loom support rod towards the front of the printer a little, as you install the filament spool on the spool holder.

After inserting the filament into the extruder, ensure the extruder is warmed up, and press the

Flush

button for that extruder.

Loading Gcode and Printing

The DX printer stores gcode files before printing them. There is no connection required between the computer and the printer, once the print begins.

There are several ways to load and print a gcode file:

1. Drag the file from a Windows folder and drop it on the



button. It will be uploaded to

the printer, and then printed.

2. Click on the Upload & Print button, and find your file in the Windows File Explorer. Select it to upload, and the print will start once it has been uploaded to the printer.
3. To upload gcode for later printing, use the  button in the same way (drag & drop, or click on it). The file will be uploaded, but not immediately printed. You will be able to then select that file from the G-Code Files list and print it.

Printing Files that have been previously uploaded to the printer

Select the Gcode Files panel by clicking on the  button in the left side button bar.

The panel will list the files that are available on the printer. For example, starting like this:

<input type="checkbox"/>	File Name	Last uploaded ↓	Object Height	Layer Height	Filament Usage	Size	Generated by
<input type="checkbox"/>	 Printable Bed Lattice (PLA).gcode	10/2/2018, 5:39:19 PM	0.25 mm	0.1 / 0.25 mm	2293.6 mm	501.6 KiB	KISSlicer - PRO

To print the file, click on the name. A confirmation popup will ask if you want to print it:



Clicking on Yes will start the print.

By default, the list of files is sorted in order the files were uploaded. This is indicated by an

arrow symbol beside the Last Uploaded header item:

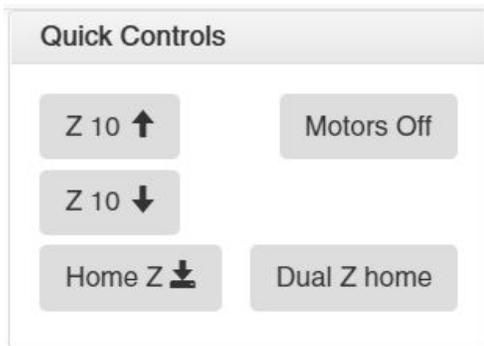


When the arrow is pointing down, the newest file will be at the top.

This order normally is desired, so that the last files to be uploaded are easy to find, when the list is very long. You can click on each header title to sort by that item. Clicking again on an item that is already the sort order will reverse the order.

Quick Controls

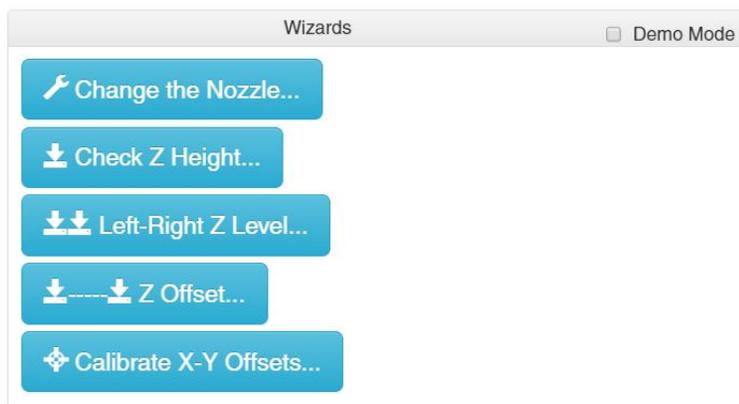
Use the buttons in the Quick Controls panel to do the everyday tasks like raising the head or homing Z.



The only one there that is “special” is the Dual Z home button. This will later be automated, but right now, it should be pressed about once every day, when the bed is clear i.e. when it is safe to Z-home. It just ensures that the bed is kept level from left to right. It is not usually at all critical to do this, and it may work just fine with a week or more between pressing that button.

Maintenance Wizards

The Wizards are provided to make many of the common maintenance tasks easier, and especially helpful when learning to do these tasks... or remembering how to do them.



Each Wizard will lead you through the selections and steps that each one involves.