You bought your Onefinity CNC, Machinist or Woodworker, and now you're trying to decide what you will use for making all those projects you have lined up. Fundamentally you have a choice between a router or a spindle.

For the router, Onefinity currently only supports the Makita RT0701c or the Bauer router from Harbor Freight. There is a Makita clone that will also work. It really makes no difference which router you put in there, as the same pros and cons will apply.

The pros are a decent price to get started. The Makita is about $100 and the Bauer can be purchased for cheaper if you use your ubiquitous coupons on one. The Makita is fairly robust and the brushes should give you between 250 and 400 hours before you have to replace them. Replacement brushes are relatively cheap. A 1/8-inch collet can also be purchased to give you a wider palette of tooling to use on your projects.

The cons of a router versus a spindle won't show up if all you use is 1/8-inch tooling or don't cut more than plywood and hardwoods. A router usually has 6 speed settings which are predetermined by the manufacturer. As chip load is determined by speed, federate, and number of flutes (cutting edges), you are limited in choosing how fast you are going to cut a material because you will only have six choices. Due to the formulas below, you will really only get to use THREE of the six settings because of chip load and ipm restrictions. For 1/8-inch end mills it is not too much of a problem, but at ¼-inch end mill cutting you will start to see a big difference.

Using a spindle you can pick whatever speed you want within the range of what the spindle is capable of producing. For example, spindles come in flavors from 0-24,000 rpm, 8000-24,000 rpm, or 9000-24,000 rpm. You can increment by 100 rpm if you see fit for greater control of your cutting tool. For the purposes of this article, I'm using a water cooled Huanyang 1.5kw 65mm spindle with VFD. It can be 110v or 220v as that does not affect the speed and feed of the spindle. This spindle uses ER11 collets of either metric or imperial sizing. The spindle used does NOT come with a minimum rpm recommendation. Be aware that several spindles do. I've seen 65mm spindles that have 8000rpm or 9000 rpm minimums.

The whole process of cutting material with your CNC comes down to math. Fortunately there are only TWO formulas that you need to deal with as a beginner, or someone who doesn't want to break bits trying to figure out that "sweet spot" of rpm, feed rate (ipm), and chip load.

Here are the formulas:
FeedRate (IPM) = RPM x # of cutting edges x chip load
Or
Chipload = Feed Rate / (RPM x # of Cutting Edges)

Both will get you to the same place, it's just a matter of which variable you are trying to solve.
With the router it is simple. We know the RPM range as there are only six settings. Three of them we can almost discard right away as the RPM speed is on the high end of usefulness for most cutting materials. Most endmills have 2 flutes so we have that value as well. Chipload is determined based on what you are cutting. There is a speed and feed calculator in the Onefinity Facebook group files and in the Forum.
In a nutshell, for a ¼-inch endmill, it goes from .009 to .011 for hardwoods, .011 to .013 for plywoods, and .013 to .016 for MDF. Notice there is some overlap between materials. I'm not covering plastics,

non-ferrous materials such as bronze, brass, and aluminum here. If you can cut those, I'm assuming you already know what you're doing.

Here are the general recommendations for various size cutting tools and materials:

Tool Dia.	Hardwood	Plywood	MDF/Particle
1/8''	.003 -.005	.004 -.006	.004 -.007
1/4''	.009 -.011	.011 - .013	.013 - .016

Hardwoods are defined as species such as oak, maple, walnut, cherry, and ash. Plywood is either the premium Baltic birch or the stuff you get from your local store.

Depth of cut is typically ½x to 1x diameter of your cutting tool. As you get better and wish to experiment, 2x diameter is possible. For example, a ¼-inch end mill will use a depth of cut of .125 inch to .250 inch. As you get better and more familiar with the material you are working with, you may be able to go to .500 inch per cut. Again, to be safe and protect your bit investment, stick with the more conservative depths of cut until you get used to your machine and materials you are cutting.

For comparison, we will use 1/8- and ¼-inch 2 flute end mills. Vbits and ballnose bits are not used because both a router and spindle will handle their speed and feed requirements, with the caveat that the spindle will give you more precise control of your cut.

For this experiment, we will use .0045 chip load for the hardwoods and as plywood falls into this range, the numbers apply for both materials.

1/8-inch 2 flute endmill:
Router
10k rpm = 90 ipm
12k rpm = 108 ipm
17k rpm = 153 ipm

A spindle gives additional control, but nothing earth shattering in terms of performance.
8k rpm = 72 ipm
9k ipm = 81 ipm

In addition, the spindle can use the router rpm rate, as well as 11k, 13k, 14k, and everything in between. This is where you get to fine tune your cutting of materials.

Where you see the difference will be in the ¼-inch endmill values. If you're cutting our worst case material – MDF – with the router you have twelve settings that range from 260 ipm to a whopping 544 ipm (17k rpm). That covers the low end of .013 chip load to the high end of .016 chip load, which the machine is not even rated to go 544 ipm!

On the spindle side of the chart, I've included three common rpm rates (7k, 8k, 9k rpm) that illustrate how much more control you'll be able to achieve without the machine flying thru its paces. You also have twelve settings shown but note the ipm difference: 182 ipm to 288 ipm using the same low end .013 chip load to the .016 chip load. 288 ipm at worst case versus 544 ipm is no comparison.

1/4" 2FL	Makita RT0701C				65mm Spindle		
	10k	12k	17k	22k	7k	8k	9k
0.009	180	216	306	396	126	144	162
0.01	200	240	340	440	140	160	180
0.011	220	264	374	484	154	176	198
0.012	240	288	408	528	168	192	216
0.013	260	312	442	572	182	208	234
0.014	280	336	476	616	196	224	252
0.015	300	360	510	660	210	240	270
0.016	320	384	544	704	224	256	288

For the Makita, I've include setting number four (22k rpm), but just look at those high feed rates (ipm). This is where it is abundantly evident where the spindle will outperform a router on feed rates alone.

I hope this short primer helps you with whatever you use: router or spindle. The key take away is that bit breakage and tool life can be maintained with either but the spindle will give you much more control and a huge variety of speeds and feed rates (ipm) that a router cannot come close to.

I've seen a number of people request a good speed and feed for cutting a material and well intentioned people responding with rpms that the Makita cannot even adhere to.

My conclusion:
If you're going to primarily use 1/8-inch end mills, get the router. Spindle won't be worth the price to upgrade, unless you're cutting a lot of MDF.
If you're going to use the ¼-inch endmills for ANYTHING but hardwood, (like MDF), then get a spindle.
If you're going to hog thru hardwoods with ¼-inch endmills, or want to use 3 flute endmills, or want to cut thru non-ferrous metals, then the spindle is your choice.
If you want greater control of your feeds and speeds, get a spindle.
If you can live with 10k, 12k, and 17k rpm, then calculate your inch per minute speed and chip load, and stick with the numbers and get a router.
If you're going to do a lot of 3d work with ballnose bits, or use v-bits a lot, then a router is fine.
If you're going to run a job that goes more than one hour, spindles will last longer due to bearing life versus brushes.
Brushes last 250 – 400 hours. Bearings of good quality will last years.
If you require high precision, get a spindle as they have less run-out.