PEAK OF FLIGHT NEWSLETTER

ISSUE 530 / SEP 15TH 2020

IN THIS ISSUE

MAKING AIRFOILED FINS ON A CNC MACHINE

https://www.apogeerockets.com/Rocket-Kits/Skill-Level-3-Model-Rocket-Kits/Johnny-Star-Commander

www.ApogeeRockets.com
4960 Northpark Dr, Colorado Springs CO 80918
Ph# 719-535-9335

APOGEE COMPONENTS

PEAK OF FLIGHT

Making Airfoiled Fins on a CNC Machine

By Pete Taran

I picked up an Aspire kit (https://www.apogeerockets.com/Rocket-Kits/Skill-Level-2-Model-Rocket-Kits/Aspire) recently and and started to think about putting it together. I read with interest in the instructions about sanding a perfect airfoil in the supplied balsa. After thinking about it, I imagined a better way using some equipment I already had on hand, namely a CNC router that would allow me to make some perfectly prepared rocket fins, each one identical in every way.

FIGURE 1: A CNC ROUTER, MARKED WITH X AND Y AXIS. THIS ARTICLE DETAILS THE PROCESS I USED TO CREATE THE FINS USING A CNC ROUTER TO CARVE REAL THREE DIMENSIONAL AIRFOILS FROM WOOD.

About CNC Routers

Computer Numerically Controlled (CNC) routers (Figure 1) have been around for quite some time. When they first made their appearance in the 1990s, they were expensive, big and complicated. Over time, they have become simplified to the point that serious hobbyists and small businesses can afford them. So what is a CNC router? It is essentially a machine with a fixed bed that has a gantry which can move a router both up (y axis) and across (x axis) the entire usable surface of the bed. Additionally, the gantry has a third axis which moves the router up and down (z axis) perpendicular to the machine bed. Various manufacturers of routers accomplish this in different ways, but most accurate small format machines like the one I use employ a zero backlash ball screw to move the gantry through the 3 planes. This is accomplished by rotating small stepper motors that rotate the screw a finite amount which is translated into linear motion by the screw and the captive nut located on the gantry or machine base. This system is very accurate, allowing repeatability to less than .001". The business end of the operation is the router which holds a cutting end mill to machine the part. Entry level CNC routers use ordinary routers of the type you might buy at Home Depot. Higher end machines use what is known as a high speed spindle which has more power, runs very quiet and has little run out in their arbor. That's about all you need to know about the machinery. The CNC router I use is made by a company called Shopbot which is a US company which makes their products on site in North Carolina.

To move from idea to cut part, you need to design the part in software. People that are familiar with 3D printing are familiar with the process to design a 3D model that the printer can utilize to render the part. A CNC router is similar to a 3D printer, the difference being a spinning router is used instead of a system to dispense liquid plastic. There are many different programs which allow you to create a cut file, but the one that many use is designed by a company called Vectric. The software is essentially two parts. The first part is a drafting like program which allows you to draw and design the part. The second part of the program allows you to create cut routines called toolpaths which tell the router what to do and how to do it. It basically is many lines of code that tell the router gantry where to move in x,y,z coordinates. There are also machine specific commands

Continued on page 3

About this Newsletter

You can subscribe to receive this e-zine FREE at the Apogee Components website www.ApogeeComponents.com, or by clicking the link here Newsletter Sign-Up

Newsletter Staff

Writer: Pete Taran
Layout / Cover Artist: Matthew Martinez
Proofreader: Michelle Mason

Making Airfoiled Fins on a CNC Machine

Continued from page 2

such as to start the spindle or define the move speed of the gantry. These commands are referred to as “G Code” and are standard in CNC programs. The software makes it easy for you to design the part, create cut routines to make the part, and then save them to a file that the CNC router can interpret and execute to cut the part.

Designing the Aspire Fin

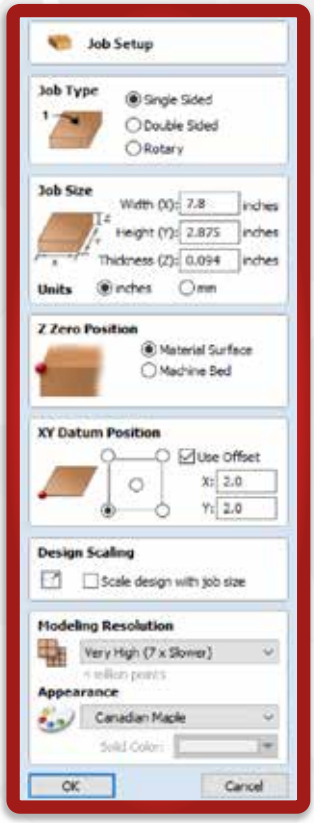


FIGURE 2: CNC SOFTWARE - JOB SETUP

Enough about the theory, let’s design the Aspire fin! The first step is to set up the job size in the software which is defined by the material you will cut the part from. In Figure 2, you can see the dialog box which is the first step in setting up a job. It’s important that you get the measurements of your stock correct, if the actual piece of wood you are using is 7.75” long and you input 8”, your part may be truncated when it’s cut. The measurements should be exact. In practice, if you make sure your stock is slightly larger than you need, you will never be caught short.

The thickness is particularly important, especially as it relates to the three dimensional type work we are working on. As mentioned above, if the file says the part should be .094” thick, and it’s really only .080”, some aspects of the part will not be cut.

The material offset allows you to move the stock you are carving from in and up on the bed so you have room to clamp it to the machine bed. Otherwise the lower left corner of the work would be right at the origin, which is the 0,0 point on the machine bed. Once the setup of the material is

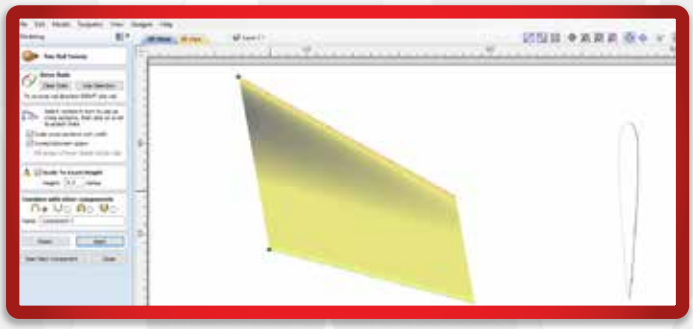


FIGURE 3: ORIGINAL FIN DRAWING IN CNC SOFTWARE

complete we can get on to drawing the part.

In this example (Figure 3), drawing the fin is a pretty straightforward task. I simply measured the balsa fin supplied in the Aspire Kit with a micrometer and replicated it in the software. In the software which I use, which is ironically called “Aspire”, each drawn line is called a vector. Typically, vectors are joined to each other to form a boundary which has no breaks in it. In this case, since we are going to build a 3 dimensional part, we won’t do this for reasons that will be clear in a moment. We also need to develop the cross section of the fin which I just did by eye to look like the upper part of an airfoil. This was pretty easily done with the ellipse tool. While the height of the elliptical airfoil doesn’t have to be exactly the same size as the height of the fin, it should be close.

You can be as precise as you need while drawing the fin. Once the drawing is complete, you can literally just hold the physical part on the computer screen to check if you

Continued on page 4

Making Airfoiled Fins on a CNC Machine

Continued from page 3

have the angles and distances correct. Adjust as needed to suit to replicate the fin you are working to create.

Once you are satisfied with the 2D look of the fin, it's time to create the actual three dimensional model that will become the fin. To do this we use a tool called the two rail sweep. In a nutshell, you define the two rails, in this case the upper and lower edge of the fin, and then sweep a profile along those rails. It's quick and easy. To do this, you click on the upper rail, and then the lower, then select "Use Selection". Last, you then click the profile of the fin that you want to use. Before clicking apply, be sure to select both scale cross sections with width and sweep between spans. The scale cross section with width is particularly important and a very nice feature. It adjusts the height of the fin profile depending on how wide the rails are at any given point. As you can see from the shaded model in Figure 4, since the fin is taller at the root, it is thicker there, and then tapers as you get to the other end which has the most narrow part of the rail. When the fin is carved, it will be thick at the root, and thin at the edge which is the most efficient design.

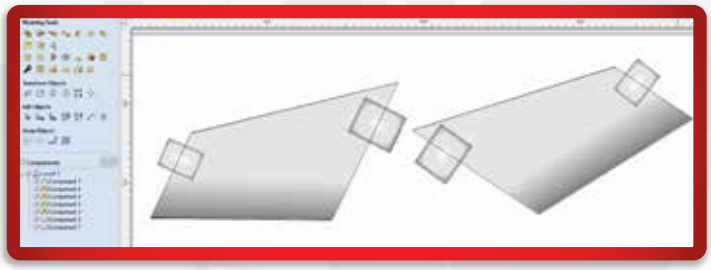


FIGURE 4: THE SHADING MARKS THE THICKER REGIONS

The last step is to copy the entire fin and replicate it. I experimented with carving a solid fin with two sides, but the trailing edge of the fin is only .020" thick and it was just too thin to leave unsupported when I flipped it to carve the other side. I decided that I would carve a left and right side and then glue them together. I realized that this idea would be good for another reason. By varying the angle of the two sides in the stock, the grain of the wood would run in opposing directions and be sort of like plywood when glued up. This makes for a dimensionally stable fin and also one that is more durable. To do this I just copied the vectors, flipped them to make a mirror image and then recreated the other side of the fin using the two rail sweeps. The last step was to create some tabs to hold the fin in the stock. The easiest place for them is at the ends where there is some thickness to the fin and also one that can be easily cleaned up with a block plane since that area is flat. Once the two halves are complete, you can rotate them at any angle you want. If I had wider stock, I would have rotated them equally so the grain on the two sides was perpendicular to each other.

The last step in the electronic part of the fin creation is to create the toolpaths that the CNC router will use to cut the part. This is done with built in wizards to create the file that is required to run on the machine. Most three dimensional type carvings are done in two steps. The first is a roughing pass which hogs 90% of the waste quickly with a regular endmill, and the second is a finishing pass which is done with a ball end mill and takes quite a bit of time and only removes a small amount of wood. However, since only a very small amount of wood is removed in this case, I opted just to create a finishing toolpath. The cutter that I use is a 1/8" ball end mill and if you remove too much stock it will not last very long.

First, select the tool and define its parameters. There are only a few fields here, but they are very important. I start with the rpm of the spindle. The Shopbot will spin at 18000 rpm, but I don't like to go to 100% of its rated capacity. In this case, I selected 14000 rpm. The feed rate is perhaps the most important value in the tool parameters. The cutters I use are made of carbide, and the manufacturer specifies a narrow range of how big a chip it cuts on each

Continued on page 5

Making Airfoiled Fins on a CNC Machine

Continued from page 4

flute with each revolution. There is a wizard where you input the RPM and the number of flutes, in this case 2 and the chip load specified by the manufacturer. For most hardwoods, .005" is a good value to use. That means that each time the cutter revolves once, the CNC router will advance the cutter sufficiently that each flute will cut a .005" chip. Once you have inputted those three parameters, the wizard gives you the feed rate. In this case for these parameters and bit, the gantry will move at 2.33 inches per second. If you are breaking bits or burning your part from excessive friction, you are moving the bit too fast or too slow. Take the time to perform this calculation, it only takes a minute. The last thing that needs to be defined is how much stepover you want the CNC router to take with each pass. A 50% stepover means that the bit will move over .0625" with each pass. That would result in a very rough carving. Remember that the end of the cutter is a ball, so in practice you can only move the cutter over a small amount unless you want a lot of clean up. This is trial and error but in general, the smaller the stepover, the better the surface finish. The downside of small stepovers is that it adds considerably to the time it takes to carve the part. Like most things in life, competing factors demand a compromise between surface finish and time. Since the fins will be carved from poplar, it will be pretty easy to clean up. I opted for 11% which is a pretty small amount but which experience shows results in a great no sand finish.

The final step in creating the toolpath is to select the 3D profile you want to carve, specify the machining limit boundary which is typically the model , and what type of cutting you want to do, climb or conventional. This is defined by if the router moves clockwise or counterclockwise in relation to the part. In most situations, climb cutting gives a better surface finish. Once complete, select calculate. The software will create the toolpath based on the parameters that you specify.

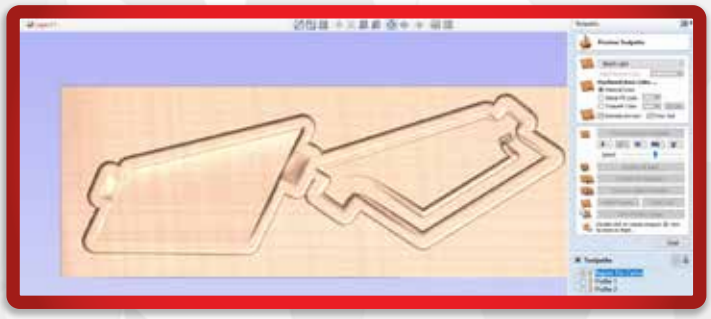


FIGURE 5: ELECTRONIC PREVIEW OF THE CUTPATH

One of the nice features of the software is that once the toolpath is calculated, you can preview it electronically without even turning on your CNC router. This is handy and saves a lot of wasted time, wood and effort. In Figure 5 you can see one fin carved with the other one in progress. The software calculates the toolpath based on the most efficient movement of the router, this is typically the one that takes the least time.

You also need to define a profile to cut through the stock and to define a clean perimeter of the fin. These are called Profile 1 and 2 in the toolpath list. Profile toolpaths are very easy to create. The router just follows the vector that defines the perimeter of the fin. You can specify it cut outside, inside or on the vector. Most uses of the profile toolpath are to free the part from the stock so outside is selected.

Continued on page 6

Making Airfoiled Fins on a CNC Machine

Continued from page 5

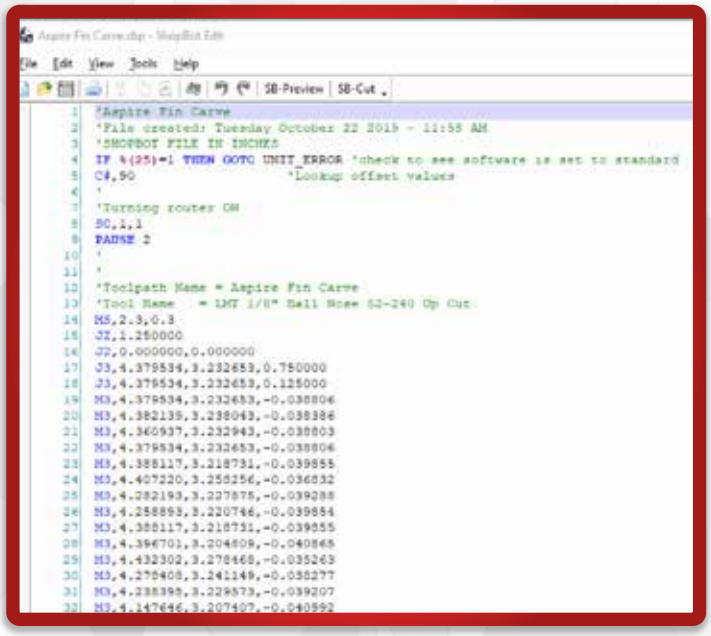


FIGURE 6: THE FIN CUT FILE

Once you are satisfied with the virtual part, it is time to translate 1 and 0s into a real part you can hold in your hands. Not to be anticlimactic, but cutting the part on the router is the easiest part of the process. It will do exactly what you told it to by way of the toolpaths. That is not to say that you don't have to sometimes tweak the toolpath by changing the feed rate or the bit stepover, but as you gain more experience with your router, it's not uncommon to go from file to a successful part on the first pass.

The toolpaths are saved to a shopbot part file, which is nothing more than a notepad file which lists the lines of locations that the router should move and how fast. Once this file is saved, the electronic part of the fin creation is complete. In Figure 6, you can see a portion of the actual file I will use to cut the fins. There are over 10,000 lines of location code for just these two small parts. The J commands jog the gantry to the location at full speed, the M commands move the cutter at the defined feed rate you specified when setting up the cutter. The locations are always shown in X,Y,Z format. You can see that once the move commands start, the Z values are negative which means the cutter is engaged with the material doing some actual cutting.

Cutting the Physical Fin

The first step in cutting your part is to mount the stock on the bed of the machine and zero the end of the bit with the material surface. There are many methods for fastening the stock to the machine, from as simple as double stick tape to as elaborate as a gasketed puck which is connected to a vacuum pump. I prefer wooden hold downs which engage slots in the table and can be tightened with an allen key. As you can see in the picture, this piece of wood is not going anywhere. For a small job like this which is removing a minimum of wood, you don't need to get too carried away with the hold down method, you just don't want it to shift. If it does, the bit will catch it and chew it up, hurling the wooden part across the room and potentially breaking the carbide end mill in the process. A bigger issue to be aware of is you want to be very certain of where your hold downs are and where your bit will be travelling. The CNC moves where you tell it to. It doesn't know there is a steel bolt in the way. If you cut into a bolt, you will see a shower of sparks just before the end mill breaks. It's not good for the spindle or any other part of the machine. Most software like Aspire lets you preview the toolpaths to ensure this doesn't happen by watching the CNC cut the part virtually as shown above. Better to find out before cutting as opposed to after you have messed up your machine.

Once you have your stock held down and are certain

Continued on page 7

Making Airfoiled Fins on a CNC Machine

Continued from page 6

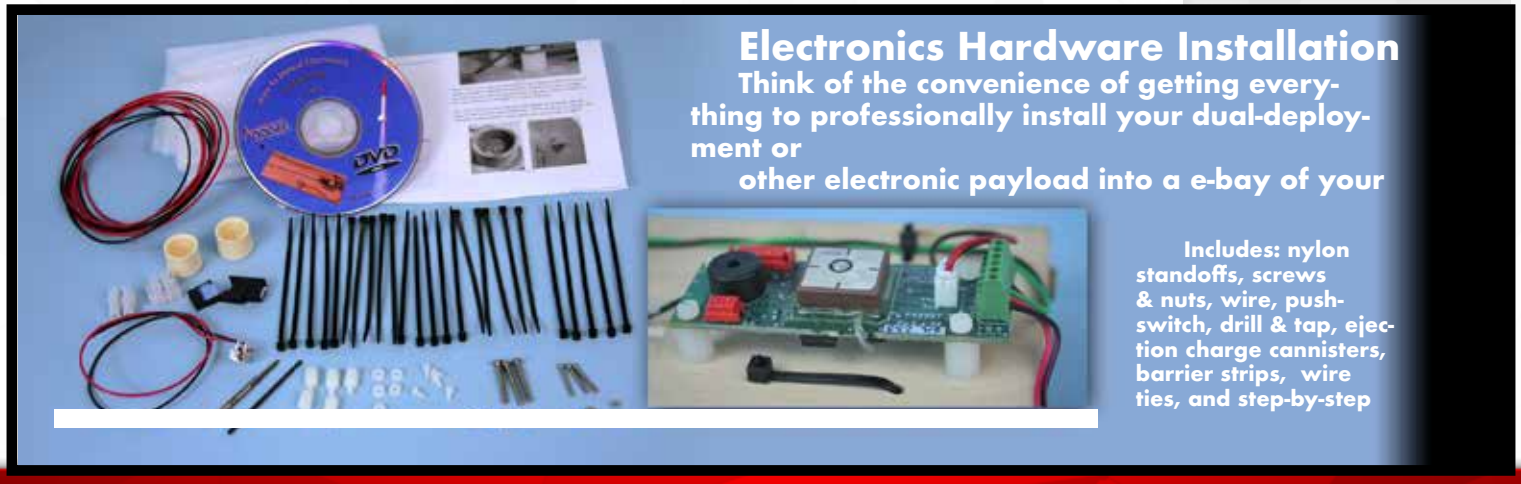
that your toolpaths are acceptable, the last step before hitting run is to zero your axis in all three directions. Most CNC machines have tiny contact switches that define the limits of travel at the origin which is the lower left corner. The machine will sequentially drive to the extreme left and extreme bottom in turn and bump against these switches. When it does, it breaks a continuity circuit and the machine records that location as “0” and then moves off the switch. The routine is automatic, most machines you tell it to zero the axis and it just does it with no further user involvement. The last step is to zero the Z axis once the bit is secured in the router or spindle. This is a little more elaborate procedure. There is an aluminum plate of known thickness that is placed on top of the stock. You then move the spindle so it is over the plate and tell the software to zero the Z axis. It will slowly lower the spindle until the bit touches the aluminum plate and it will then stop, retract slightly, and repeat it a second time. When it stops the last time it records this as “0” in the Z axis. The software automatically subtracts the thickness of the aluminum zero plate to get a precise setting. You are now finally ready to hit run!

FIGURE 7: THE FRESHLY CUT FIN HALVES GLUING TOGETHER AND HELD TOGETHER WITH CLOTHES PINS

Cutting the part is easy. Select the cut file with the machine controller software and hit “cut part”. The bit will leave the origin, or return to it first and then move to the part and begin the cut routine. As can be seen in Figure 6, it calculates the fastest way to cut the part which may not seem logical but is always the quickest. Most machines have a provision for dust collection which is a good idea as CNC routers make a lot of dust and chips. The boot is very effective in vacuuming up the chips as they are carved. When the routine is complete, the spindle returns to the origin and shuts itself off. The finished part can be seen in Figure 8.

The fins at this point are completed. The only task that remains is the mundane job of freeing the halves from the stock and gluing the two sides together. Some care must be taken to cut the fins from the stock which they were carved from. The trailing edges are very thin and fragile

Continued on page 8



Making Airfoiled Fins on a CNC Machine

Continued from page 7

until they are glued to their mating half. An ordinary Xacto knife works well. If you can, cut slightly proud of the carved edge. While one would think that wood glue would be the perfect adhesive for gluing the halves together, I discovered that the very thin sections of the fins can warp and twist because of the water present in the glue. Your preferred brand of two part epoxy is the best choice. Since you are gluing long grain to long grain, almost any variety will work. I used 5 minute epoxy for this job. Just put a very thin coat on each side, align the two halves and then clamp the perimeter with old fashioned wooden clothes pins as shown in Figure 6. When the epoxy has completely hardened, you can sand the edges smooth or even better, use a sharp block plane to make both halves perfectly mated. The leading edge of the airfoil will need a very small amount of sanding and it will be perfect. Once all the fins are glued and you are satisfied with your work, glue them to your rocket as you would any fin. I used 15 minute epoxy and once cured I followed that up with a very small amount of RocketPoxy (https://www.apogeerockets.com/Building_Supplies/Adhesives/G5000_RocketPoxy_8_oz_Package). Since the fins are surface mounted, that really helps hold them on the rocket. Be sure to rough up the glassine layer of your tube with some 320 grit sandpaper to give the various epoxies something to stick to.

You could literally write volumes about various approaches to CNC machining. This is an overview type treatment to show the steps involved from project ideation to completion. While it might seem like this would take days to complete, once you become skilled in the software and running the machine, the entire process shown here can be completed in less time than it takes to watch a movie. The best part is if you ever need to make another set, just clamp the stock to the machine and fire it up. You can make as many as you want, all identical and all perfect in every way. For perspective, the total machining time of one sheet like the one shown is 11 minutes and 45 seconds. You can make it much faster, but that comes with more sanding. Just very light sanding is required at the settings described in this article.

If you don't have the space or resource for a CNC of your own, all is not lost. If you live in an area that has a maker space, many have state of the art CNC routers which you can use after a short training session. You of course are responsible to design the part and arrive with the toolpath on a zip drive that you can use. The Aspire program that I use lets you save the toolpaths in just about any format you might need for any machine. You just need to specify it when you save the toolpath. Additionally, most of these maker spaces have a PC with the design software on site, so you can arrive, design your part and cut it all on maker space supplied equipment. This is a great way to decide if CNC routing is for you before deciding to buy a machine of your own.

FIGURE 8: THE COMPLETED ASPIRE ROCKET WITH AIRFOILED FINS