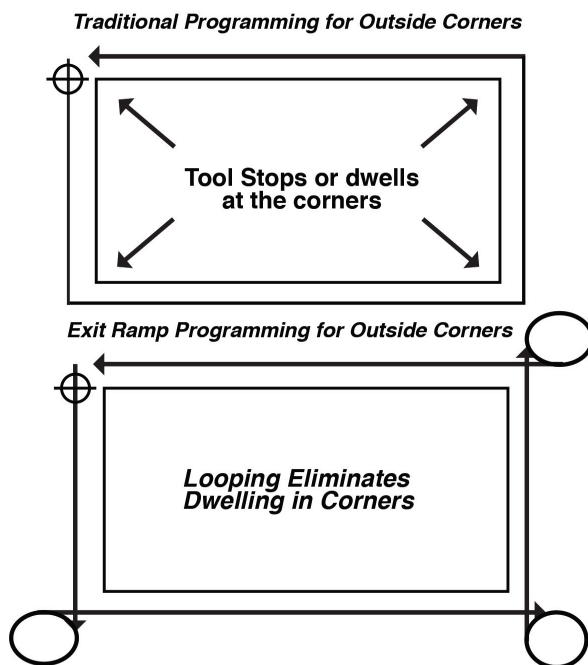


# Fixturing and Routing of Plastics with CNC

With the ever increasing use of routers to machine plastics, there has been a leap forward in the design of tooling to produce high quality finishes on a variety of products. High-speed steel, carbide tipped, and solid carbide tools have been manufactured in a variety of geometries and sizes to rout most plastics. Once an optimal bit is selected, however, achieving good productivity still involves determining appropriate programming methods, part fixturing, and proper spindle speeds and feed rates.

## Programming Techniques

The goals of routing wood and routing plastics are the same, high quality finishes at fast feed rates commensurately many of the programming techniques that apply to wood routing will work well in the plastics industry. The primary difference in plastics routing is the ability of cut chips to re-weld them to the finished surface. In softer plastics this can occur frequently and lead to a poor edge finish. Preventing this re-welding and producing a smooth edge finish while attaining fast feed rates is the secret to productive plastics machining.



The key to preventing chips from re-welding is simple - keep them cool. The easiest method is fast feed rate. Due to programming limitations, this is not always practical. Most routers have acceleration, deceleration, and curve speed limitations when cutting radii and corners. Dead stops should be avoided whenever possible. When cutting outside corners, the router will stop and dwell while changing directions. At 18,000 RPM, a double edge tool will contact the part 600 times per second and generate significant amounts of heat. This heat will not only increase the instances of chip re-welding, but will also decrease the tool life and raise tooling costs. A solution to tool dwelling is to utilize "exit ramp" programming. By programming corners as outside loops, the tool is not allowed to dwell and you can achieve a square corner.

A second case when router bits dwell is during the initial plunge of an inside cut. As the bit is boring, it is continually re-contacting the cut surface and unless the bit is a spiral or has shear, the chips are not being evacuated. This generates heat both from the rubbing and from the fact that the bit must do extra work re-cutting chips that remain in the hole. "Ramp in" cutting can eliminate this effect by gradually plunging the bit (z-axis movement) as it begins its forward travel (y- or x-axis movement). If needed the bit can travel backward after the full depth of plunge to eliminate the cut ramp. The dwell time involved with this reverse traverse is significantly less than that involving a straight plunge and rout operation.

A final suggestion to reduce dwell time: when boring a dedicated hole, actually rout the hole. Using a small diameter bit, ramp into the hole in a circular fashion and use a routing action to cut the hole to size. This allows you to hold tight tolerances and prevents the occasional blow-out on the underside of a hole when the plug is ejected.

While ramp programming to remove dwell time may seem to increase the routing path, the higher production feed rates that are attainable along with the increased tool life should make the operation economically attractive.

Assuming no software restrictions, if chip re-welding is still a problem after removing dwell points, move from HSS tooling to solid carbide as this will enable an increase in feed rate. Increasing feed rate, as stated before, can greatly reduce the instances of chip re-welding.

Sometimes because of part configuration, thickness, or composition, it is difficult to produce a high quality edge on a finished part. From a programming standpoint, there are techniques that can be used to increase finished edge quality. A rough cut and finishing pass combination work well on many thicker plastics. By leaving approximately .080" on the edge with a roughing tool, a finishing tool can clean up the edge and have enough material to cut so that the tool remains stable and does not begin to chatter. An added benefit is that the pieces produced per finishing tool are greatly increased while the more durable roughing tool is the one subjected to increased wear.

When cutting nested or mirrored parts with a single pass, operators may notice a decrease in surface finish on one of the exposed edges. Frequently, surface finish can change depending on whether the tool is presented to the material in a climb cut configuration or a conventional cut configuration. Generally, conventional cutting yields a better edge, unless a finish pass is used, in which case the second pass can be a climb cut. If nested part cutting does yield problems, the cut can be accomplished in two passes with a smaller diameter tool. The first pass will finish cut one side, and then the tool travel will be reversed and the remaining side will be finish cut.

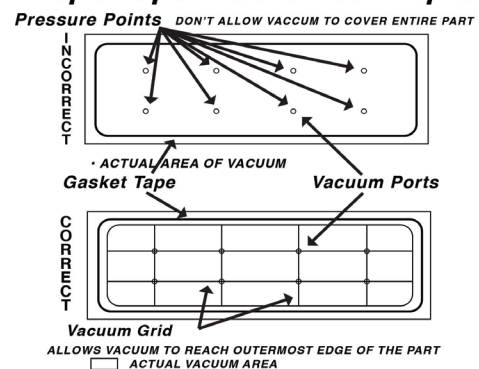
Finally, when cutting laminated plastics or products that have an abrasive layer, tool oscillation can greatly increase tool life. Materials such as plastic laminated with aluminum can cause a severe wear line on both carbide and high-speed steel. By oscillating the tool vertically (z-axis) during the cut, this wear can be spread over a larger area and allow the bit to continue cutting before it is dull.

## Fixturing

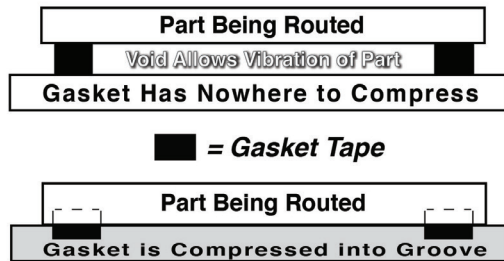
Quality production demands quality material, quality tooling and quality fixturing. Fixturing must be solid and reliable. Anything else will ultimately lead to poor edge finish and reduced tool life or broken tools. That said, there are specific techniques and configurations that can lead to a more efficient and practical hold-down system.

Vacuum hold down is the most prevalent method in the CNC industry today and it is important to get the most out of the system. First of all a piece of MDF with weather-stripping tape and a few holes drilled in it is NOT adequate. Vacuum hold down with a spoil board is capable of extremely rigid part fixturing, but only if utilized correctly.

### Proper Spoilboard Techniques



Using the router to create a grid connecting the vacuum ports allows the vacuum to reach all edges of the part to be machined. This will increase the holding power of vacuum system and allow better edge finishes due to a rigid holding configuration. Using proper gasketing tape in an oversized channel will also increase the lifetime of the spoil board. If the tape used is not for gasketing applications and has "memory", it will not expand back to its original state after repeated compressions and the vacuum system may begin to bleed off. Additionally, if the channel is not oversized, when the tape is compressed by the part it will have nowhere to go. This may prevent the part from contacting the vacuum surface and allow vibration to occur.



Other improvements for spoil boards include building dedicated boards for particular parts. One example involves cutting parts that have small scrap pieces. When the finished part is cut, excess material (outside corners, plugs from boring, etc.) can become missiles if they are too small to be held effectively by the vacuum pressure. As they chatter on the table they can contact the router bit and either cause

damage to the bit or be "shot" off the table. To eliminate this problem, build up the spoil board in certain areas and seal the edges so that the part is actually being held on the top of a pedestal or plateau. In this configuration, the excess material will fall to the main spoil board when cut and be clear of the cutting tool.

Dedicated spoil boards can also be useful when material composition demands a down cut spiral or shear tool. Soft plastics require that the chips be cleared quickly and aggressively. When using a down cut bit without a raised spoil board, the chips are not able to clear out of the cut. By routing channels in the spoil board below the areas to be cut, it is possible to give the chips a place to clear.

If these configurations still do not provide sufficient holding force and safety, the parts can be held with riveted tabs or screwed into the spoil board through the center of scrap portions. This is a last resort due to the fact that setup time per piece is increased and throughput is reduced.

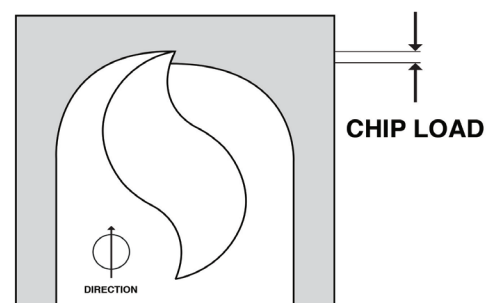
## Speeds and Feeds

If the part to be machined is fixtured securely and the correct tool has been selected for the material, spindle speed and feed rate will be the determining factors on the finished quality of the part. Speeds and feeds can vary greatly depending on router horsepower, tooling, and part composition; however, it is possible to make an educated guess at the correct ratios and to then fine tune the finish.

The defining ratio of speed and feed combinations is "chipload". Chipload is the thickness of the chip that is removed by a cutting edge per revolution. Expressed mathematically:

$$\text{Chipload} = (\text{Feed rate in IPM}) / ((\text{RPM}) \times (\text{\#of Flutes}))$$

In effect, increasing the chipload will cause a larger chip to be removed. The larger the chip removed, the more heat that is removed with it, and the longer the tool life. The primary means of increasing chipload is to increase the feed



rate as this has the added benefit of increasing the parts produced per hour. Chipload can also be increased by lowering spindle speed if feed rate is already at a maximum. Decreased chipload, means the number of times that a cutting edge is presented to the work piece is increased. Every router bit edge has only a finite number of times it can be used to cut before it is considered dull; therefore, the highest chipload that will produce an acceptable finish should be used to prolong cutter life.

Since CNC operators do not think in terms of chiploads, but rather speeds and feeds, it is useful to have some "rules of thumb" when determining rates. For the following examples, a spindle speed of 18,000RPM is assumed. For soft plastics, solid carbide spiral tools that have geometry specifically for cutting that type of plastic can be run at approximately 300 ipm. Solid carbide "O" flutes should also be run that fast in order to clear the chips. If finish begins to degrade, the spindle speed can be increased in order to maintain the same production rate. High-speed steel "O" flute tools require slower feed rates in order to prevent the bit from deflecting and causing chatter or "knife" marks.

Harder plastics work well with low-helix tools that have been designed to break the plastic chips away cleanly. These tools can be run around 300 ipm. Double edged "V" flute tools can run anywhere from 125 ipm to 250 ipm depending on style and bit composition and also produce an excellent finish. It is important to understand that in all cases, whether routing hard or soft plastics, chips (not dust) must be made. Large chips will not re-weld to a cut surface and will prolong the life of the tool. If the cut waste that is produced is dust, that means the chips have been re-cut numerous times or the chipload is too low. The tool life will suffer as well as the edge finish.

Fiber reinforced plastics are different from other types of plastics in that it is very difficult to determine the type of chip being produced. Because of the structure of materials such as fiberglass, aramid, and carbon fiber compounds, chips are not formed during the cutting process. In these instances, it is best to run the bit as fast as possible. The cooler the bit is when finished, the longer the tool life expected of the bit.

If, despite adjusting speeds and feeds, the best cut still produces a hot tool or causes occasional chip re-welding, forced air can be used to evacuate the chips. First make sure the dust collection system is operating efficiently. Then, air forced through a directional nozzle can be used to clear the chips. Additionally, several companies manufacture Venturi effect nozzles which can drop the temperature of the air charge and provide additional cooling as well as chip evacuation.

With the ever increasing formulations of plastic in the marketplace, there is going to be a continuing need for high quality machining and finishing work. After proper bit selection, the most essential items to successful routing of these materials involve optimum programming techniques, solid fixturing, and fast speeds and feeds. Be sure as much emphasis is placed on the tooling, fixturing, and programming as is placed on the CNC equipment that is expected to utilize it.

Van Niser  
For more information  
about the company's  
cutting tools,  
call (800) 234-1560 or  
visit [www.onsrud.com](http://www.onsrud.com)