

CNC ROUTERS

SELECTING THE RIGHT TOOL, MAXIMIZING MACHINE TIME,
FINISH, TOOL LIFE & PROFITS



6201 Fort Worth Avenue
Fort Worth, TX 76112

817-461-7171 Metro

Visit us at easaw.com



I am a torque wrench—use me



*I am a collet—I am inexpensive but critical
and need to be replaced every 400 to 600
hours or when damaged*



Table of Contents

TOPIC	Pages
<u><i>Introduction</i></u>	3
<u><i>Factors to consider before selecting a tool</i></u>	3-4
<u><i>The many factors affecting tool life and finish quality</i></u>	4 -6
<u><i>Collets & Tool Holders</i></u>	6-7
<u><i>Tightening Stands & Torque specifications</i></u>	8-9
<u><i>Torque Wrenches & Accessories</i></u>	9-10
<u><i>Taper Wipers</i></u>	10
<u><i>Spoil Boards, Hold-Downs & Dust Collection</i></u>	11-13
<u><i>Techniks Aggregate Heads</i></u>	13-14
<u><i>Climb vs. Conventional cut</i></u>	15
<u><i>Tool Materials</i></u>	15-16
<u><i>Tool Geometry</i></u>	16-19
<u><i>Chipload</i></u>	20-21
<u><i>Feed & Speed</i></u>	21-22
<u><i>Things to Avoid</i></u>	22-23
<u><i>Index</i></u>	24-25

Introduction:

The following are general guidelines for profitably operating CNC routers when machining wood, plastic, composites, and other man-made materials. There are many factors to consider and the choices can sometimes seem overwhelming. The purpose of this manual is to familiarize you with some basic principles that will enable you to adapt to any cutting situation.

Friction during the cutting process results in enormous heat generation. In metalworking, you may consult a machinist manual and find out exactly what tool to use and how fast to run it—the heat can be removed using flood coolants and lubricants. The heat produced when cutting materials such as wood or plastic isn't handled quite as easily. ***This is perhaps the most important concept for you to understand—heat is your enemy; it dulls tools and fatigues metal.*** When cutting wood, plastics, etc., you control heat through tool geometry, dust collection, and manipulating the chipload.¹

Each routing situation is different and adjustments should be made to achieve the desired goal. It is a continuing trial, error, and improvement process. Through proper setup, machine maintenance, tool selection, and manipulating the feed (rate of travel) and speed (RPM of the spindle) you can keep costs low while maintaining quality and production.

When cutting through 3/4" material at rates in excess of 1000 inches per minute (IPM) you are moving a lot of material quickly. Take a minute and think about what you are asking this machine to do. Most of the time you are using a tool that is only 1/4" to 1/2" in diameter. The spindle rates are up to 24,000 RPM. If everything is not in perfect harmony you are setting yourself up for failure.

¹ In the simplest terms, chipload refers to the size of the chip produced during the cutting process. Assuming a 1 flute tool, the chipload would be equal to the distance of travel during 1 revolution of the spindle. See sections on chipload and feed & speed.

Many factors need to be considered when evaluating any cutting situation, before selecting a tool:

- **What is the desired finish?** Chip-free, hidden part, some minor chips ok, etc. Are both sides (top, bottom) of material of equal importance? The desired finish will often dictate not only what type of tool we can use but also how fast we can cut and whether we must make multiple passes.
- **Which is more important, speed or finish?** While striving for both, sometimes we must assign priorities or make compromises. Higher quality finish *can* mean higher machining costs (multiple passes, use of more expensive tooling, slower feed rates, etc.). However, ironically, most people run their tools at too slow a feed rate at too high of a RPM, resulting in heat buildup and premature tool wear.
- **What are the machines capabilities and limitations?** It is important to know your CNC's capabilities and limitations. If your machine's top feed speed is 600 IPM as opposed to a machine capable of feed speeds of 1200 IPM, your tool selection and setup will be quite different. It is important to note here that just because your machine is capable of running at 3400 IPM does not mean you can cut at that rate. When programming any cut for the first time, it is important to ascertain at what feed that part is to be cut. Part size or geometry alone may preclude you from running more than a few hundred IPM.
- **Do you have sufficient ability to hold the material you are cutting?** If you can't hold it, you can't cut it. This is one of the most common problems that I encounter. I am continually amazed how many people will accept minimal feed rates and quality because of chatter and movement. Your ability to make a clean fast cut is related to your ability to hold the material firmly without vibration. Insufficient hold-down can be responsible for everything from premature tool wear and breakage to sub-standard parts, chatter marks, chipping, and material movement. A little planning and effort will give you huge payoffs in productivity and cost savings. When purchasing a CNC router, don't skimp on the vacuum system. I have never met anyone who said they had too much.
- **What is the material you are cutting?** Raw particle board, plywood, MDF, single-sided laminate, double-sided laminate, etc. Different materials have different cutting properties and may restrict your tool selection and will often dictate geometries, feed, speed, etc.
- **Are there any operations after this one that will cover, conceal, or change the shape of the part being cut?** For example, the top edge will be rounded over or the edge will be covered in such a manner so that a small chip would be of no importance.
- **Is exact size important?** Does another part rely on the accuracy of this part? Maintaining exacting measurements may require multiple passes after a tool is serviced or if the size is non-standard. [Edge of Arlington](#) offers insert and diamond tooling designed to solve these types of problems.
- **Part configuration and size are important considerations.** Small parts can be hard to hold. Intricate parts, parts with holes, curves, and short cuts can be challenging due to heat generated because of a

machinery's inability to make instantaneous speed and directional changes. Advances in machine capabilities have greatly improved the speeds associated with travel time and the ability to change direction. However, a bit turning at 18,000 RPM still turns at 300 inches per second and a good Boy Scout can start a campfire with a stick at a much lower RPM.

To maximize quality and production time, it is important to understand how the many factors that affect tool life and finish work together:

- **Programming.** Good programming techniques can make all the difference. If true optimization is to be achieved, programmers must be familiar with their machine, material, software, tooling, capabilities, limitations, and idiosyncrasies. They must also be in close contact with the machine operator and open to feedback. Using entrance and exit ramps, feed and speed variations, directional changes, selection of tool path, etc., the programmer can compensate for machine limitations and utilize machine capabilities to their fullest. As technology advances, software and machinery manufacturers are continually upgrading and addressing some of these issues. There is no substitute however for hands-on experience and trial and error. Programmers should not be afraid to experiment and should always document their successes and failures.
- **Operator.** A well-trained, experienced, observant operator can be invaluable. If quality and efficiency are to be achieved, the operator must be in close contact with the programmer. The operator is the last line of defense. An operator should be instructed in the safe operation of the machine, and to report problems such as heat buildup, burning, part movement, chatter, and inappropriate tool wear and maintenance issues. They must have the proper tools to allow them to do their job—tightening stands, torque wrenches, well-maintained machines, and access to information. This is an area where many people try to skimp. They spend hundreds of thousands on machinery but won't spend \$300 for a torque wrench, \$30 for a collet, or a few hours training for the operator. They want to make it an "idiot proof," minimum wage, automated job—not a practical solution unless you wish to micro-manage and are prepared to pay the high costs associated with premature tool and machine wear and tear.
- **Vibration.** Vibration is a major source of tool wear and poor finish quality. Factors affecting vibration include how securely the material is held; the type, thickness, and rigidity of material to be cut; collet condition, collet torque, spindle bearings, machine condition, and chipload.
- **Machinery.** Horsepower, rigidity, run-out, spindle condition, clamping system, etc. A well-maintained machine also includes the vacuum and dust collection systems. Preventive maintenance is a necessity to maintain quality finished parts; avoid major, expensive repairs; and maximize machine time, tool life, and profits. Your machinery dealer can be a valuable partner when it comes to machinery selection, questions, and issues.

- **Collet.** Your router bit is only as good as the collet it is in. This inexpensive, wearable part is often overlooked and can lead to poor finish, premature tool wear, tool breakage, and unnecessary machine wear and tear. The collet must be in pristine condition and under the proper torque to align the bit perfectly on center—ALWAYS USE A TORQUE WRENCH. The torque wrench should be used for tightening only, as using it for loosening can lead to miscalibration. A tool with run-out will cause inconsistent chipload, creating finish degradation, heat buildup, and tool damage. Collets should be kept clean, replaced regularly, and never over tightened. An improperly colleted bit can cut the tool life in half, or more.
- **Heat.** Heat buildup is the biggest cause of tool wear and breakage. Coolants are not practical for most CNC operations involving wood or composites. However, use of air cooling and air flow from dust collection can significantly reduce heat and extend tool life. In woodworking and plastics, heat is controlled by manipulation of the chip and dust collection. Adjust the feed and speed to achieve the biggest chipload possible, while producing the desired finish. **The chip takes the heat from the tool; the heat dulls the tool.** Never leave a tool in the cut longer than necessary as heat will build very rapidly. Heat is a function of surface footage per unit of time. The denser the material, the faster the feed rate needed to minimize heat buildup.
- **Dust collection.** This system removes the chip from the cutting surface—the chip takes the heat with it. The importance of a good vacuum system cannot be overstated. If waste material is not removed from the cut, the chips may be repeatedly pulled into the cutting circle. This can cause the cutting life of the tool to be reduced by as much as 50% and the finish may take on a glazed or burnished appearance. The flow of air over the spindle and tool also helps to keep the bit cool. If dust collection is not working at top efficiency, the heat can quickly damage the edges of the tool. Dust can also become a health and safety issue. Dust control is a major problem in most manufacturing situations. Upshear spirals can aid in chip removal but put upward pressure on the material.
- **Tool sharpness.** As a tool dulls, the edges begin to "pit." In carbide tooling this is caused more from heat and chemicals in the materials being cut than from the actual cutting process. Tools should be changed at the first sign of deterioration. Using a tool for too long can cause finish degradation of the product, tool damage, or breakage. Cutting that "one last part" may result in an excessive "dull line" or damage. You may not be able to sharpen the tool, or sharpening may be limited as it becomes necessary to grind off more of the carbide to restore the cutting edge.
- **Tool type.** High speed steel (HSS), carbide-tipped (CT), solid carbide (SC), diamond (PCD), etc. The type of material to be cut will often limit, if not dictate, what type of tool to use. HSS works well on soft material. PCD is very wear resistant but is very brittle and expensive. For a more detailed discussion read the section on tool types.
- **Tool geometry.** Different materials require different hook angles, relief angles, clearance angles, etc. When the tool is sharpened, it is important that these angles be maintained. When a tool is serviced the dimensions will change—most often the diameter and clearance angles will become smaller. After a tool has been serviced, adjustments in feed rates and tool off-sets may be necessary.

- **Feed & Speed.** These two factors go hand in hand. When adjusting one you must always consider the other. The feed and speed should be adjusted in such a way that there is a constant and even pressure during cutting. Too fast a feed rate given hold-downs and other factors will cause strain and deflection of the bit (run-out). Too slow a feed rate will cause friction and burning. At the proper feed and speed the router bit will be at or near room temperature—*never touch a tool when it is turning!* For a more detailed discussion, see the sections on chipload and on feed & speeds.
- **Material to be cut.** Most man-made materials are hard on tooling; they can contain everything from chemicals and glue to sand and metal. All of these are sources of tool wear and breakage. Even natural woods have chemicals, knots, different grain directions, and foreign matter. The properties of the material to be cut will dictate what type of geometry will give you the best finish. Also, the thicker the material, the longer the tool, and therefore the greater chance of run-out and deflection.
- **Laminates.** Laminates are made for looks, strength, and scratch resistance. The same processes that make them scratch resistant, also makes them harder and more costly to machine. Laminates can be extremely dense and can contain practically anything. They are notorious for dulling tools quickly and for chipping or fraying on the edge. Often a router bit may be chipping the laminate but the rest of the tool will be sharp. Oscillating the bit up and down during the cutting process is one way that machinery manufacturers and software companies are trying to combat these type problems. When cutting laminates, controlling heat through proper feed, speed, and tool selection is the key.
- **Drilling.** Most of the newer machines have drilling heads, utilizing the European style boring tools. These tools generally are 57mm or 70mm long and have a 10mm shank with a flat and set screw in the bottom of the shank for minor height adjustments. The boring heads work off a gear system and therefore alternate right hand (RH) and left hand (LH) rotation. The bits and heads are usually color coded—black for RH rotation, red or orange for LH rotation. They are available in brad point (BP) for general purpose, thru-hole (TH) for use when going all the way through the material, and hinge style for larger holes. You may also use the router for drilling, but you must be able to turn the RPM down to very low RPM's (2500 or less) to really control the heat and chip. In either case, the use of router bits to drill will result in excessive heat and should be used only when there is no other option. You will dull a router bit more drilling one hole than cutting out the entire part.

The CNC router has evolved to be the new workhorse for the 21st century. In the last 20 years or so the CNC router has replaced the table saw, panel saw, and hand-held router as the tool of choice. One router and a good operator can produce what it took a small army to make only a few years ago. As more and more shops embrace the CNC and the technology becomes more advanced and affordable, the need for information and understanding becomes ever more critical.



Collets & Tool Holders:

One of the most critical and most overlooked parts of the CNC router is collet and tool holder maintenance issues. Collets and tool holders are critical parts of any CNC. They should be thoroughly cleaned and inspected during every tool change.

Collets are inexpensive and expendable, especially compared to the cost of the tooling that relies on its accuracy. [Edge of Arlington](#) carries [Techniks](#) collets and holders and parts. Guaranteed accurate.

- **Collets are made from spring steel** and should be replaced at least every six months of daily usage (400-600 hours of run time), or when there is any sign of wear or damage.
- **Tools should be tightened with the aid of a torque wrench.** Over-tightening can cause distortion and run-out, leading to tool wear, breakage, and finish degradation. Under-tightening can lead to bit slippage and twisting in the collet. Torque wrenches should only be used to tighten—using a torque wrench to loosen a nut can lead to miscalibration.
- **When using full grip collets**, whenever possible the tools should be chucked to fill 80% of the depth of the collet. Fillers are available to prevent collet collapse in the event the 80% rule cannot be maintained. Full grip collets have slits running from the top and bottom.
- **Never chuck up on the cutting edge of the tool**, always just below the fade of the flute.
- **The tool holder is also subject to deterioration** and should be inspected regularly for wear and damage.
- **Collet brushes are available for cleaning.** Clean using a nylon or brass bristle brush only. Use alcohol after cleaning to ensure there are no chemical residues remaining.
- **Spindle wipers are available for cleaning** the spindle where the tool holder mounts.
- **Recommended torque is based on the style** of collet used in the tool holder. An HSK tool holder for example is available with several collet options. Torque has been calculated based on the unique properties of each collet style.
- **It is important to use a setup fixture** or similar device to safely hold the tool holder while changing the bit.
- **Damage to the tool, holder, or operator** can occur if the holder is dropped or improperly clamped.

Cutting tools last longer when you... Torque it right!



Over-tightening can distort collets, causing runout problems and premature failure of the collet, tool holder, and cutting tool.

Improperly tightened tool holders are a safety hazard. Always use a tightening stand and torque wrench to tighten collet nuts and pull studs.



Benchtop Mount Tightening Fixtures

Part No.	Type
7852	BT 30
7854	BT 40
7856	BT 50
7870	CAT 40
7872	CAT 50

Collet Nut Torque Specifications

Collet Nuts	Wrench Type	Torque
ER 16	Slotted	42 ft/lbs
ER 16	Hex	42 ft/lbs
ER 20	Slotted	59 ft/lbs
ER 20	Hex	59 ft/lbs
ER 25	Slotted	77 ft/lbs
ER 25 Mini Nut	Castle	28 ft/lbs
ER 32	Slotted	100 ft/lbs
ER 40	Slotted	130 ft/lbs
SYOZ 25/TG 100	Hook	90 ft/lbs
TG 75	Hex	90 ft/lbs
TG 100	Hex	90 ft/lbs

Retention Knobs Torque Specifications

Retention Knobs	Wrench Flats	Drive	Torque
All 40 Taper	0.750"	3/8"	76 ft/lbs
CAT 50 Taper	1.250"	1/2"	99 ft/lbs
BT 30 Taper	0.535"	1/4"	36 ft/lbs
BT 50 Taper	1.187"	1/2"	100 ft/lbs
ISO 20 Taper	0.346"	3/8"	20 ft/lbs
ISO 30 Taper	0.535"	3/8"	36 ft/lbs

HSK Style

BT, CAT Style



Top or Side Mount Tightening Fixtures

Part No.	Type
17883	BT 30, CAT 30
7884	BT 40, CAT 40
7885	BT 50, CAT 50
17893	HSK 63A, 63B, 63D

Side Mount Tightening Fixture

- For CAT, BT, HSK, ISO spindles
- Change Retention Knobs from underneath
- Bolts right to workbench

Part No.	Description
NTS-40 Taper	CAT 40 or BT 40 taper
NTS-BT30-38	BT 30 taper - 38mm flats
NTS-BT30-41	BT 30 taper - 41mm flats
NTS-BT30-Slot	BT 30 with drive slot
NTS-ISO25-27	ISO 25 - 27mm flats
NTS-ISO30-36	ISO 30 - 36mm flats
NTS-ISO30-38	ISO 30 - 38mm flats
NTS-ISO30-47	ISO 30 - 47mm flats
NTS-HSK63F-36	HSK 63F - 36mm flats
NTS-HSK63F-38	HSK 63F - 38mm flats
NTS-HSK63F-41	HSK 63F - 41mm flats
NTS-HSK63F-46	HSK 63F - 46mm flats





ER Castle

ER Slotted

ER32 Closed

Hex Type

TG Hook

DNA ER 16

Retention Knob

Fixed Head

Part No.	Collet Nut Size	Nut Type	Max. Torque*
04576	ER 11 Mini	Mini	12 ft/lbs.
04577	ER 16 Mini	Mini	20 ft/lbs.
04578	ER 20 Mini	Mini	22 ft/lbs.
04587	ER 25 Mini	Mini	28 ft/lbs.
04580-16	ER 16 Slotted	Slotted	42 ft/lbs.
04601-16	ER 16 Hex	Hex	42 ft/lbs.
04580-20	ER 20 Slotted	Slotted	59 ft/lbs.
04602-20	ER 20 Hex	Hex	59 ft/lbs.
04603-25	ER 25 Slotted	Slotted	95 ft/lbs.
04604-32	ER 32 Slotted	Slotted	100 ft/lbs.
04605-40	ER 40 Slotted	Slotted	130 ft/lbs.
03690-25	SYOZ 25/TG 100/RDO 35	Hook	90 ft/lbs.
04018-TWA	TG 75 Slotted	Hook	90 ft/lbs.
DNA16-TWA	DNA 16	Slotted	42 ft/lbs.
04588-R	ER 32 Slotted	Slotted / Closed	100 ft/lbs.
SD-TWA	SYOZ-20	Hook	40 ft/lbs.



Real lambskin provides the best cleaning. Use with any type of machine with a taper of 7/24 CAT, BT or DIN 69871.

Spindle Taper Wipers

- For CAT, BT, HSK spindles
- Remove contaminants to improve T.I.R.
- Keeps spindles and tool holders clean



Spindle Taper Wipers

Part No.	Taper	Total	Taper
17706	30	170mm	60mm
17707	40	188mm	78mm
17708	50	240mm	120mm
7710	HSK 63A	165mm	31mm
7711	HSK 63F	165mm	31mm

Order online at eoasaw.com

Phone 817-461-7171 Toll Free (888) 461-7171

Fax 817-795-6651

Failure to properly maintain these parts can and will lead to expensive machinery and spindle repairs, not to mention higher tool and machining costs.



Another critical area involves spoil boards, hold-downs, and dust collection:

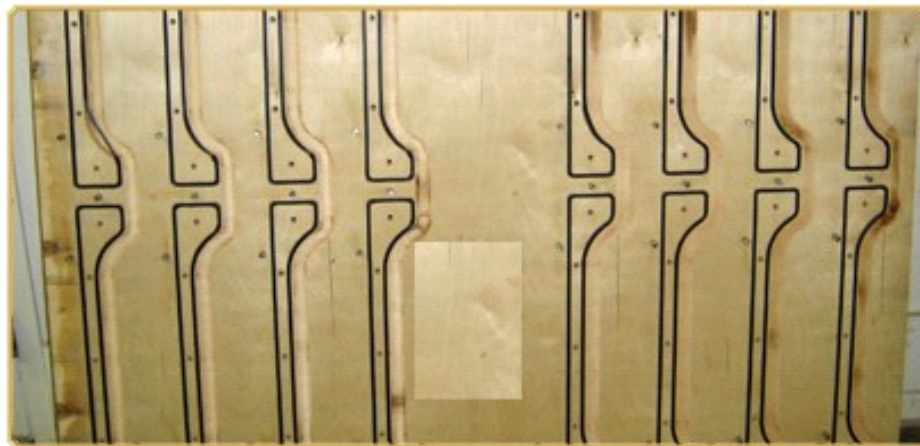
Holding the part to be cut firmly and without movement or vibration is often the biggest challenge faced when cutting on a CNC. Chatter, movement, premature tool wear, tool breakage, poor edge finish, and chipping are the result of inadequate hold-down. How well you can hold the part is perhaps the biggest factor in how fast you can cut. There is a lot of information on building dedicated spoil boards and jigs so we will only cover the basics.

Some factors to consider include:

- **Some CNC operations are best suited to using flow through** or suck through vacuum systems. Particularly well-suited for cutting large parts and sheets of material, these types of routers utilize replaceable spoil boards—so called because they are underneath the material to be cut and are cut and “spoiled” during machining. The spoil board can then be resurfaced and used again, sparing the router table underneath from damage. These types of systems are easy to use and require minimal maintenance. Using low (LDF) or medium density fiberboard (MDF), both sides are machined level using a large “fly” cutter to remove any imperfections or coatings. Vacuum is evenly distributed to the entire surface of the table. Air loss from the sides can be minimized by using rubberized or latex paint. Areas not to be used should be sealed with sheets of laminate, plastic, or other non-porous materials. Grooves resulting from the cutting process can cause vacuum leakage when cutting a variety of patterns. Resurfacing the spoil board should be done regularly.
- **Smaller or more intricate parts are better suited** to dedicated spoil boards due to vibrations and part movements. Cutting small parts can be challenging since a larger volume of air is lost due to the number of channels cut between parts. As more parts are cut, more air pressure is lost, resulting in vibration and part movement.
- **Leaving a thin skin (onion skin) or programming tabs** at certain points can help to hold small parts together but require a secondary routing or sanding operation.
- **Use dedicated spoil boards for recurring part runs.** Taking the time to make a proper spoil board will pay for itself by allowing faster feed rates, giving you a better finish, and prolonging tool life.
- **Weather stripping is not gasket tape!** The repeated compression and stress the tape is subjected to requires a more flexible, crush-resistant material. A quick Google or other search engine query will provide you with numerous sources.
- **Gasket tape should be applied in a channel** to establish the outside perimeter of the part to be cut. The channel should be roughly half the thickness of the tape with a 1/16” side clearance to allow for

compression. Without a channel, the tape will simply compress and lose its memory, allowing the part to vibrate and resulting in wear and finish degradation.

- **Drilling a few holes in a piece of MDF** or particle board is not a proper spoil board. This technique may suffice in some instances—particularly large, thick pieces—but is not satisfactory in most situations and will contribute to tool wear and finish degradation. Simple holes create pressure points and allow parts to move, vibrate, and flex.
- **Grooving the interior area of a dedicated spoil board** will allow the vacuum to flow evenly to all areas of the part to be cut. Holes should be drilled inside the channels of the vacuum grid, allowing for a more even vacuum.
- **When designing a dedicated spoil board**, don't be afraid to be creative. If the material is thin and wants to flex, a combination of grooves and “supporting walls” can work very well.



- **Deeper grooves along the router path** can be used to give chips a place to go when using a downshear geometry.
- **Plateaus for the part to rest on** (sealed along the edges) can allow small scrap parts to fall safely to the spoil board instead of becoming projectiles. There are as many variations as there are part configurations. The main goal is to allow vacuum to flow to all corners of the part and at the same time give support where needed to minimize part movement and vibration.
- **Pods are great for large pieces** and when you need access to the underside of a piece, such as putting a bullnose on the edge or doing horizontal drilling on a point to point. Pods are not as well-suited to manufacturing small parts and “cookie” cutting. In these instances, you may be able to adapt your pod system to use a spoil board—see your machinery dealer.
- **Most spoil boards are 3/4” thick.** If you prefer a thicker spoil board, use a product such as Sierra Pine’s 2” thick LDF.
- **Most manufacturers recommend 38-45 lb MDF.**
- **The higher the CFM the better** for holding small parts. When working with a pump with low CFM’s a thicker spoil board (1-1/4” to 2” thick) may work better on small parts because although it allows less air to flow, it also lets less air escape. A high CFM pump would be one that ran 450 CFM and above.
- **Consult your machinery dealer** regarding other hold-down options and for technical assistance.

CNC SPOILBOARD SURFACING



Part Number	Large Diameter	Shank Size	# of Wings
STRAIGHT CUT			
SB25-2	2 1/2"	1/2"	2
SB40-2	4"	3/4"	2
SB40-3	4"	3/4"	3

UP SHEAR CUT			
SBU25-2	2 1/2"	1/2"	2
SBU40-3	4"	3/4"	3

CARBIDE INSERTS			
INDEXABLE - 4 CUTTING EDGES			
Part Number	Description	Size	
SB-Insert	10 Pc. Insert Pak	14 mm x 14mm x 2mm	
SB Screw	Replacement Insert Screw	M5x7	
SB Wrench	Insert Wrench	T-15 Torx	

- **Some machinery dealers offer roller-hold-downs.** These are particularly useful when you are dealing with materials that are subject to warping. Warped material can be difficult to cut and may cause damage to tooling as material stress is released during cutting. A roller-hold-down by itself is not usually sufficient to give you a quality finish. A roller-hold-down should be used in conjunction with a good vacuum and spoil board system.
- **A properly maintained dust collector is important** to remove the dust (and with it heat). Much of the dust generated can be a health hazard and should not be inhaled. Consult the manufacturers MDS sheets for chemical contents.
- **Overly long bristles** can cause small parts to move and should be trimmed with a pair of scissors.
- **Filters should be inspected** and/or cleaned daily.
- **Check pressure gauges** and make a note of what they should read. If you are not sure, consult your machinery dealer or manual.

Techniks Aggregate Heads

Today's CNC can do just about anything. [Techniks](#) has a line of Aggregate heads that will enable you to chuck up just about any tool.



Quattro: 4-drive output head



Duo: 2-drive output head



Add sawing capability to your CNC

Our aggregate heads outperform the competition. We guarantee it.

We can say this because our aggregate heads are designed by ex-Benz engineers, who knew the weak points of the Benz design and corrected them. Only the best components, manufactured to the highest quality, make it into our aggregate heads. All our parts are painstakingly checked for quality.

For example, our bevel gears are subject to a *90-point inspection* that measures the gear's overall 3D shape, the taper of the teeth, and the consistency of the tooth spacing and angle. *Our standard for gear tolerance is 1/200th of a millimeter.* Our competitor's is .5mm.

Our manufacturing process makes the difference.

Each of our aggregate heads is individually hand-assembled by a highly skilled technician, who is personally responsible for build quality. That's right; *one craftsman does the whole job* from start to finish.

Once the assembly is complete, the technician performs a test run of the aggregate for 30 minutes. After the post-run inspection and measurement, the technician finishes sealing the aggregate and *signs his name to the inspection report* verifying that it has passed his personal inspection.



Climb cut vs. conventional cut:

Climb and conventional cutting refer to the direction of travel in relation to spindle rotation. In the case of a right-hand rotation spindle, climb cutting (clockwise direction) can produce a different result than conventional cutting (counterclockwise direction).

- **In many circumstances conventional (CCW) cutting** will provide the best finish, especially in plastics and with larger diameter tools. If the scrap piece has a better finish than the finished part, a change in cutting direction will provide you with the opposite results. Don't be afraid to experiment with cutting directions or have different cutting directions on the same workpiece.
- **There are also left-handed tools and holders** available for left-hand rotation—climb cutting would then become counter-clockwise and conventional cutting would become clockwise.
- **When cutting the interior of a part** as opposed to the outside perimeter, conventional cut becomes climb cut and vice versa.
- **When cutting two parts with one pass** such as when cutting parts that are nested or mirrored, keep in mind that one side of the cut will be conventional cut and the other will be climb cut.
- **Climb cutting is more aggressive** and can move small parts.
- **When cutting hard materials** and metals such as aluminum, climb cutting is preferred.

Tool Materials:

- **High speed steel (HSS)**. Used primarily on soft woods such as pine, aluminum, and soft plastics. HSS can be sharpened to a very sharp edge and at the appropriate hook and clearance angles “slices” through soft material, reducing pre-splitting. HSS is more impact resistant than carbide but is not wear resistant enough to hold up to many of today's man-made materials.
- **Tangtung (TG)**. A cast alloy material with a high tungsten content. Tangtung retains its hardness at very high temperatures, is more shock resistant than carbide, and resists corrosion from wood acids. Tangtung, like HSS, can be sharpened to a fine edge and is primarily used in hardwoods for a superior cross-grain cut. TG can and has been used in insert tooling for CNC applications. Freeborn Tool has a line of insert shaper cutters available in Tangtung. Tangtung is *never* recommended for manmade fiber boards or plywood.
- **Carbide-tipped (CT)**. Used primarily on hard woods, laminates, composite materials, and for abrasive or resin-based products. Carbide is welded or brazed onto a steel body and profiled to the desired pattern; therefore, it is the most common type of material used for profile and custom tooling. There are many different grades of carbide. The harder the grade of carbide, the more likely it is to break during the heating and cooling process in manufacturing—problems overcome with insert tooling.
- **Solid carbide (SC)**. Used primarily on hard woods, wood composites, laminates, plastics, aluminum, and other composite materials for longer life and faster feed rates. Solid carbide can be cast into just about any shape. Solid Carbide rods are well-suited for making router bits with spiral geometries and

small form or profile tools. Solid carbide blanks are also well-suited for insert tooling and are available in many standard sizes and configurations. Solid carbide is very wear resistant but not impact resistant—strain and deflection often lead to tool breakage.

- **Diamond (PCD).** Polycrystalline diamond is primarily used on composite materials such as melamine. PCD diamond is extremely heat and wear resistant (100+ times carbide) but is not impact resistant, and in fact, it chips fairly easily. Materials must be free of all foreign matter and be homogenous in nature or tool damage may result. Machine condition and maintenance are vital to cost effectiveness. High initial cost but very low operating cost *can* make diamond tooling a very economical alternative. Diamond wafers are brazed on to a steel body and are not suited to small sizes.
- **Insert tooling.** Insert tooling can provide consistency in production. Using replaceable inserts instead of brazed-on-carbide allows you to maintain a constant diameter. Insert tooling is also able to utilize harder, more wear resistant carbide than traditional CT tooling. While very versatile in nature, the design does not lend itself well to small diameters due to the need to affix the insert with Gibbs and/or screws.

Tool Geometry:

There are a lot of different geometries involved in making a router bit—clearance angles, grinding angles, hook angles, axle angles, side clearance angles, radial clearance angles, and side rake. For the purposes of our discussion, we will only talk about the basics.

- **Tool size is a determining factor** in how fast you can cut and what types of geometries and clearance angles are possible—all major factors in productivity, finish, and consistency of parts.
- **For stability, choose the shortest cutting length possible.** You should have sufficient shank to fill 80% of the collet. Longer router bits will deflect more—run-out is more exaggerated the longer the tool.
- **The larger the diameter of the tool the faster you can go**—up to a point. While a 3/4" diameter bit will be able to handle a much larger chipload than a 1/8" diameter bit, larger bits require slower speeds for safety reasons. The following are general safety *guidelines*:
 - Up to 1" Diameter—Maximum of 24,000 RPM
 - 1" to 1-1/2" Diameter—Maximum of 18,000 RPM
 - 1-1/2" to 2-1/4" Diameter—Maximum of 16,000 RPM
 - 2-1/4" to 3" Diameter—Maximum of 14,000 RPM
 - 3" to 3-1/2" Diameter—Maximum of 12,000 RPM
- **Use the largest shank possible.** However, cutting diameters smaller than the shank diameter cause stress points at the transition and can contribute to breakage. Solid carbide tools larger than 1/2" in diameter can be expensive and are designed when extra-long lengths are needed.

The following are safety *guidelines* with relation to shank size and head diameter:

- 1/2" diameter shank—Maximum large diameter is 2"
- 1/2" diameter shank—Maximum small diameter is 1/2"
- 1/2" diameter shank—Maximum overall length is 4"

- 5/8" diameter shank—Maximum large diameter is 3"
- 5/8" diameter shank—Maximum small diameter is 5/8"
- 5/8" diameter shank—Maximum overall length is 4-1/2"

- 3/4" diameter shank—Maximum large diameter is 3-1/4"
- 3/4" diameter shank—Maximum small diameter is 5/8"
- 3/4" diameter shank—Maximum overall length is 5"



• **Straight flute tools are most often found in hand routing operations.** They can also be especially useful in CNC applications, especially when you are *not* using laminates and you cannot take advantage of the fast speeds possible with spiral tooling. Straight flute tools have a neutral effect on cutting pressure (neither up nor down) and they may chip laminates. Straight flute tools can be useful when cutting thin materials and plastics where lifting of the part is a problem. Straight flute tools can be “V-” or “O-” flute design (look down the face of the tool—an O-flute has a half moon appearance and a V-flute looks like a “V”). Hard, rigid plastics cut well with straight V-flute tools, soft plastics with O-flute tools. Brazed tools can also be supplied with a carbide boring point for use when plunging.



- **Shear / helix CT router bits can be downshear or upshear or both.** These are not spiral bits but modified straight bits. Downshear bits are the most common and usually have only a slight angle, three degrees or so. This design allows for a chip-free cut on the top of the material (bottom if upshear) and is frequently found in the manufacture of shelving using hand routers and the routing of dadoes and other grooving applications. Edge of Arlington is a distributor of “Her-Saf” style cutters, which are probably the best-known example of this type bit, and they have developed a complete line of reverse helix, screw-on cutters in both CT and insert designs that provide an exceptional finish when cutting dadoes.



- **Upcut spirals will aid in fast chip removal.** When cutting all the way through the material, the upcut will produce a **smooth finish on the bottom** but may fray or chip on the top, especially on entry into the material. Upcut spirals are preferable when grooving, slotting, or when fast chip removal is required, such as cutting soft plastics. Upcut spirals will try to pull the material from the table so a good hold-down system is particularly important. Upcut spirals are not recommended when stacking material due to the tendency of the material to move.

- **Downcut spirals will force chips downward so a good dust collection is especially important.** Downcut spirals can generally run at faster feed rates due to the downward pressure, which helps to hold the material in place. When cutting all the way through the material, downcut spirals will produce a **smooth finish on the top** but may fray or chip laminate on the bottom—back support such as a bleeder board will help to minimize tear-out. Downcut spirals are often used to cut dadoes and rabbits when a clean cut on top is needed. Downcut spirals are preferred for hand-held operations, when material is being stacked, and when cutting small pieces. Downcut will tend to try and push the material around the table so a good hold-down system is still especially important.



- **Compression spirals take advantage of both the upcut and downcut spiral geometry.** When cutting all the way through the material the compression is designed to give you a **smooth finish on both the top and bottom**. The upcut portion is on the end, which can cause tear-out on entry into the material. The up and down design tends to reduce pressures during the cutting process, allowing for faster feed rates. Because the shank side of the bit is comprised of the downcut spiral, chips are not removed as efficiently as the upcut spirals—a good dust collector is very important.
- **Compression spirals—Mortise cut.** This has become a very popular variation of the standard



compression spiral. This type of compression bit has a short upshear and long downshear. Generally, the upshear is equal to the diameter of the tool; however, it is also available in shorter lengths (mortise cut). This design allows you to use the same bit to cut through the material (taking advantage on the upshear and downshear) and make a dado cut (taking advantage of the downshear) without the need for a tool change. The short upshear is also useful when cutting materials of various thicknesses without making a tool change. The longer downshear tends to put downward pressure on the part allowing for faster feed rates.

- **Single flute router bits are often used when heat is a problem** due to the inability to run fast, such as routing holes. The single flute takes a larger “bite” than a multiple flute router bit at the same feed and speed, giving you a larger chipload and therefore removing more heat. They are also appropriate on small diameters where breakage may be a problem.
- **Double flute router bits are the most common.** They may give you a better finish than a single flute router bit at similar feeds and speeds and will give you a satisfactory finish in most cutting situations.
- **Three- and four-flute router bits are used for very fast speeds** and some finishing operations. Older model machines and intricate parts may make it impossible to fully utilize this type of bit. The multiple flutes require fast speeds to produce a chip large enough to remove the heat from the material being cut. The three-flute geometry is found in many finishing tools—designed to remove a small amount of material (.015 to .080) after a rough cut or hogging operation. Taking too small of a finish cut can cause finish degradation as the bit skips across the surface because it doesn’t have enough material to bite into.
- **Chip-breaker spirals and rougher/hoggers** are regular spiral bits that have “grooves” or “slots” cut into the flutes at staggered positions. These “grooves” act as chip-breakers that allow you to run at higher feed speeds and are primarily used in roughing or hogging operations. Chip-breakers can also help to reduce pre-splitting when cutting across the grain.



- **O-flute tools have a rounded, half-moon appearance** as you look down the face of the flute. This geometry is most often used in machining aluminum and plastics and can be made of HSS or SC. The rounded flute allows for better chip removal, which is especially important in preventing “chip-welding” when machining soft plastics such as ABS. O-flute tools can have spiral or straight geometries—straight O-flute geometries are commonly used in air routers on most types of plastics.

Chipload:

To get the most out of your CNC you must have a firm understanding of chipload. It is the main factor over which you have direct control.

- **In the simplest terms, chipload refers to the size of the chip** produced during machining. The chip size is controlled by the how fast the spindle is turning (RPM) and how fast the router is moving (Feed Rate). Assuming a 1-flute tool, the chipload would be equal to the distance of travel during one revolution of the spindle. For a 2-flute tool, the chipload would be equal to 1/2 the distance traveled in one revolution.
- **A tool provides only so many cuts; make the most out of them.** If the dust collection is not functioning properly and the waste material is not removed from the cut, the chips may be repeatedly pulled into the cutting circle. This can cause the cutting life of the tool to be reduced by as much as 50%.
- **You can use the following formula** to determine your chipload. Or if you know what chipload is desired you can work backwards to figure feed and speeds required to reach your target. A difference of as little as .001" in chipload can mean a dramatic difference to the bottom line.
 - $\text{Chipload} = \text{Feed rate in IPM} / (\text{RPM} \times \#\text{Flutes})$
 - $\text{Feed Rate in IPM} = \text{RPM} \times \#\text{Flutes} \times \text{Chipload}$
 - $\text{Speed (RPM)} = \text{Feed Rate in IPM} / (\#\text{Flutes} \times \text{Chipload})$
- **Recommended chipload varies from material to material.** A general rule of thumb is that the harder, denser the material, the smaller the chipload. On the other hand, **heat is a function of surface footage per unit of time—the denser the material, the faster the feed rate needed to minimize heat buildup.** A delicate balancing act to be sure. Variations in RPM and feed rate can produce dramatically different results.
 - Natural Woods—.012" to .025"
 - Plywood with Veneers—.003" to .015"
 - Masonite—.003" to .015"
 - Laminated Plastics—.003" to .008"
 - Acrylics—.001 to .005"
 - Particle Board—.010" to .025"
- **You may have noticed that the above ranges are large.** You can cut at a higher feed rate (chipload) when using a spiral router bit than a straight flute bit and higher still with a compression bit. Also, the better your holding ability the larger the chipload possible.
- **Don't be afraid to push it a little.** Pick a starting point and adjust feed rate up from there. At some point the chipload will become too great and the part will begin to move or the finish to degrade as the bit begins to deflect.

- **Sometimes it is easier to change the RPM.** Adjust at 10% increments until you minimize the heat, then use the above formulas to recalculate feed and speed to maximize machine time. Change only one variable at a time and observe the results.
- **Chipload is important** because the chip takes the heat away from the tool and heat is the biggest cause of tool wear and failure.
- **Keep in mind that other factors are working against you** as you try to determine the proper chipload. Plunging, deceleration at corners, small parts, holes and cut-outs, and other factors will contribute to heat buildup.
- **Some fibrous materials simply will not produce a chip**, but only fine dust. In these instances, go as fast as you can. Good dust collection will help, and sometimes air cooling can be utilized to improve efficiency.
- **When determining the proper feed and speed it is important to ascertain if the machine is achieving the programmed speeds.** You may program at 900 IPM but if you are cutting small or intricate parts it is doubtful that you will ever achieve your programmed speed. In this circumstance you can control heat by decreasing RPM or changing the type/geometry of the tool.
- **Sometimes it is necessary to make compromises.** Find the weakest link and work from there.

Feeds & Speeds:

You should keep in mind the other factors we discussed as we talk about feeds and speeds, in particular your hold-down ability.

- **At the proper feed and speed**, the chipload should be sufficient to remove the heat. In most circumstances, the tool will be at or near room temperature immediately after cutting. Burning or heat marks on the tool or the part are indications that you are running at too high RPM for the programmed feed rate.
- **The following is a starting point only**—adjust feed and/or speed in 5% to 10% increments and observe the results, then fine tune in smaller increments until you find “the sweet spot.” **The general rule is that the faster a bit is fed, the longer the life of the tool.**
- **Feed rate is the controlling factor** in productivity, so the ultimate goal is to maximize feed rate. Push the feed rates to the limits of finish and tool integrity for optimum profits.
- **The following calculations** are based on 18,000 RPM using a 1/2" diameter router bit on 3/4" material. The feed rate is expressed in inches per minute. For other diameters and thicker materials use the following multipliers.
 - 1/4" diameter tools—reduce feed by 40%.
 - 3/8" diameter tools—reduce feed by 20%.
 - 5/8" diameter tools—increase feed by 20%.
 - 3/4" diameter tools—increase feed by 40%.

- **When cutting thicker material**, decrease feed rate by 25% for each multiple of 3/4" (1-1/2" reduce 25%; 2-1/4" reduce 50%).
- **Multiple passes may also be necessary on thicker materials.** The general rule of thumb is to *make multiple passes when the depth of cut is more than three times the cutting diameter for larger tools and two times the diameter for smaller tools.*
- **Feed rate has a linear relation** to spindle speed (RPM). Assuming you are running at optimal speed, the higher the RPM the faster the feed rate possible (10% RPM increase = increase feed by 10%; 20% RPM reduction = reduce feed by 20%). This means that if you have an appropriate chipload and the tool is running cool and you increase the RPM by 10%, you can increase the feed rate by 10% while maintaining the same chipload.
- **If you know the feed rate you want to run**, use the chipload formula to determine what your RPM and chipload would have to be to achieve your goal. There are four variables involved in the calculation—feed rate in IPM, spindle speed in RPM, the number of flutes in the tool, and the chipload. If you can assign numbers for three of the four variables, you can solve the equation to find the other.

Tool type	Soft Wood	Hardwood	PB & Plywood	MDF
Carbide-tipped Straight	250 IPM	300 IPM	350 IPM	400 IPM
1F Upcut <i>Or</i> Downcut	200 IPM	300 IPM	350 IPM	400 IPM
2F Upcut <i>Or</i> Downcut	250 IPM	350 IPM	400 IPM	500 IPM
2F Up-Down Compression	350 IPM	400 IPM	500 IPM	600 IPM
3F Up-Down Compression	450 IPM	750 IPM	800 IPM	800 IPM

- **While the above feed rates are a safe starting point**, much higher feed rates can and have been achieved.
- **More detailed charts are available** over the internet from tooling manufacturers. While charts are useful to an extent—they give you a starting reference—there is no substitution for trial, error, and experience.

Things to Avoid:

- **Dead stops, also known as “dwelling in the cut,”** can lead to premature tool wear and failure. When changing directions of travel or when cutting small pieces, a one second stop at 18,000 RPM generates enormous heat and no means of extraction. Imagine rubbing your hands together 300 times per second—a sure way to form a blister. This is more of a machinery and software issue. Certain programming techniques can help to compensate, such as ramping, use of loops at sharp outside corners, and the use of exit ramps.

- **Plunge cutting**, while this is the oldest and most widely used method of entering the material, it is not always the most efficient. Plunge cutting can lead to chips wrapping around the router bit after repeated plunges. Whenever possible you should Ramp into the cut. This eliminates the heat generated during the plunging process and can eliminate burn marks at the plunge point. Plunging can also cause the bit to “walk” because there is no centering point—this will be evident as ovals holes and entry points larger than the diameter of the tool. If you cannot ramp in, a good alternative is to plunge down outside of the workpiece and enter from the side. Spiraling in can also help. Whatever it takes to keep the router bit moving.
- **Routing holes is never the optimum choice.** Use drills for drilling, routers for routing. Drills turn at much lower RPM than routers do. Router bits generate too much heat when drilling. Drilling just a couple of holes per sheet will significantly reduce the life of your tooling. In fact, you will probably dull more drilling the two holes than cutting the entire piece. If you do not have drill heads to take advantage of European type boring bits, turn the RPM down as low as possible, 4-5000 RPM at most, and plunge as fast as possible. An upshear will help to remove the chips if hold-down is not a problem. On deep holes “pecking” can sometimes help.
- **Small parts and scrap pieces** can become projectiles if left after the cutting process without sufficient hold-down. They can also be sucked into the dust collector and cause a blockage. Skin cutting and the use of tabs can help to hold small parts.
- **Over-tightening of the collet** and collet nut is a common mistake. Most operators assume that the tighter the better—this is not true and can lead to premature tool and collet wear and tool breakage. Collets are the most misunderstood and overlooked part of the CNC, often times the cause of poor finish, short tool life, and unnecessary machine wear. Collets are made of spring steel and are subject to distortion when put under undue pressure. It is highly recommended that collets be tightened to manufacturers’ specifications with the use of a torque wrench.
- **Fires—yes it happens.** A tool left in the same place for a sufficient amount of time can and will start a fire. It is important never to leave a CNC running without an operator. You wouldn’t believe how fast things can go wrong. The dust collection system and the vacuum system will aid in the fires speedy travel. If this ever happens to you, beware of flash-back when the vacuum is turned off. When it is safe to do so, it is important to remove and examine the bleeder board and table underneath, as embers can remain and once again flair up.

Understanding the different variables involved in CNC routing can be overwhelming, especially to anyone new to CNC’s. That is why Edge of Arlington Saw & Tool is here to help you in selecting the proper tool for the job. Don’t let the complexities involved keep you from maximizing tool life, machine time, and profits—help is only a phone call way.

Please let me know if you have any additions, corrections or contributions to the above material that you would like to share with others. Information presented here has been gathered over many years from various sources.

The preceding is based on 20+ years of experience helping customers to achieve their goals. It is intended as a guide only. Thanks to the many mentors that I have had over the years, from customer to vendors. There are helpful articles and lots of useful information available online. From trade magazines to tooling manufacturers, I encourage you to seek out more information. Don't settle for broken and burned tooling, take the time to do it right—even after 20 years I am still learning.

Thom Snellings

President

[Edge of Arlington Saw & Tool, Inc.](#)



Office (817) 461-7171



Cell (817) 919-7914



Fax (817) 795-6651



THOM@EOASAW.COM



[Facebook](#)



[Our Blog](#)

INDEX

Topic	Pages
Aggregate Heads	13-14
Bit Speed Guidelines	16
Bit Shank Diameter Guidelines	16
Chip-Breaker Spirals and Rougher/Hoggers	19
Chipload	3, 20-21
Climb Cut vs. Conventional Cut	15
Collets	2, 5, 6, 7
Collet Nut Torque Specifications	8
Compression Spirals	18
Compression Spirals-Mortise Cut	18
Double Flute Router Bits	19
Down-Cut Spirals	18
Drilling	6
Dust Collection	5, 11, 13
Feed and Speed	6, 21-22
Heat	3, 5
Hold-Down for Materials	4, 11, 13
Machinery	5
Materials Being Cut	6
Man-made	6
Laminates	6
O-Flute Tools	19
Operator Responsibilities	4

Programming Necessities	4
Safety	
Dust	5, 13
Set-up Fixture	7, 8
Bit Speed	16
Bit Diameter	16
Things to Avoid	22-23
Retention Knobs Torque Specifications	8
Shear/Helix CT Router Bits	17
Single Flute Router Bits	19
Spindle Taper Wipers	7, 10
Spoil boards	11-13
Straight Flute tools	16, 17
Things to Avoid	22-23
Three- and Four-Flute Router Bits	19
Tightening Fixtures	8, 9, 10
Tool Holder	6, 7
Tool Selection	3
Sharpness	5
Type	5
Geometry	6, 16
Tool Materials	15, 16
Torque wrench	2, 5, 7, 8
Up-Cut Spirals	17, 18
Vibration	5