GDP TOOLING

CNC Tooling Guide

Easy to Follow Guide to Maximize Tool Longevity, Performance, and Obtain Best Finish

www.gdptooling.com

1.800.544.8436 | info@gdptooling.com

About GDP Tooling

GDP Tooling offers a complete program of state of the art precision cutting tools. Our program includes tooling for:

- CNC Machines
- Saws
- Edgebanders
- Moulders
- Tenoners
- Shapers
- Boring Machines
- Other Woodworking Machinery

Our extensive program encompasses GUHDO saw blades and our GDP brand tooling solutions in solid carbide, carbide insert, and diamond tooling options. Tool clamping systems, aggregate heads, and other CNC accessories round out a comprehensive tooling program.

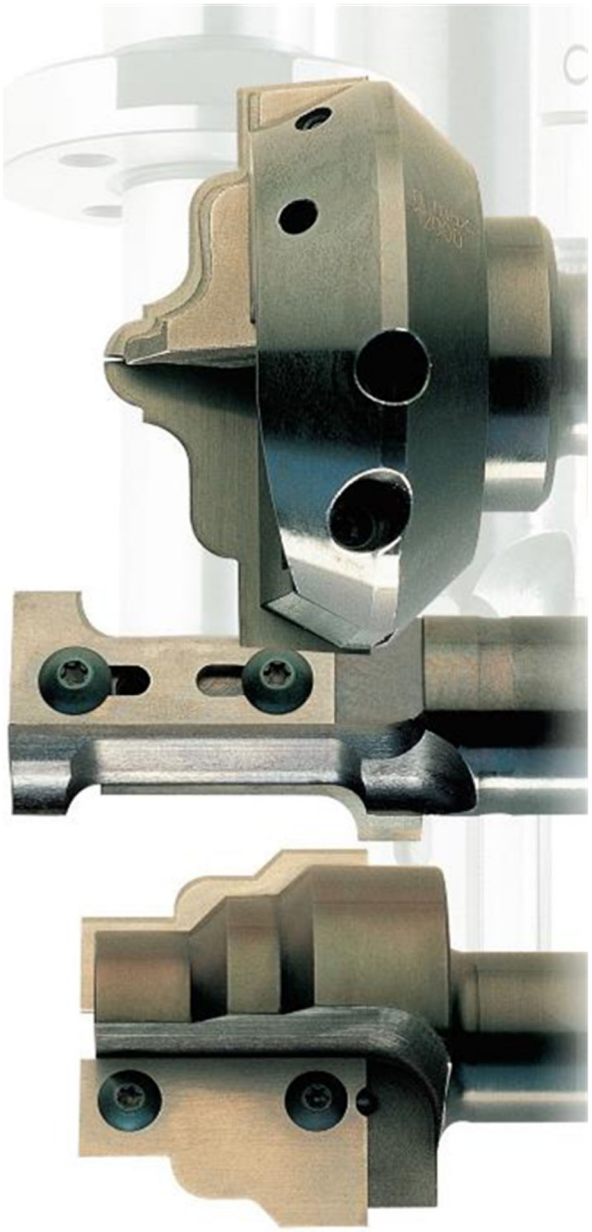
For More Information Contact:

1.800.544.8436
info@gdptooling.com
www.gdptooling.com

Table of Contents

Introduction

MAKING THE RIGHT CHOICE



Precision cutting tools and clamping systems offer a high degree of technical sophistication. There are important characteristics they must possess which provide the basis for safe, smooth, and efficient operation of a CNC machine.

Beginning with design, construction, engineering, expertise, to the balancing and inspection of the finished product, the selection of tool and clamping system made can have a profound impact on performance and production cost—with an even greater potential impact on your bottom line.

With the many options and suppliers to choose from, it does require a bit of research to make the right tooling decisions. Surprisingly, cutting tools are often the last thing thought about when buying a CNC machine, but consider this: **without your cutting tool, your machine won't produce a single thing!**

A SYNERGY AT WORK

Cutting tool performance, i.e. tool life and surface finish, is the result of a synergy between cutting tool, clamping system, material hold-down, efficiency of dust extraction, correct machining parameters (i.e. spindle speed and speed rate), and the composition of the material being machined.

For the best performance, all facets need to be right.

THE ANALOGY

Keeping in mind that a high class, high performance automobile requires high quality, precision-balanced tires to run smoothly, the same applies to cutting tools and tool holders used on a CNC machine.

REMEMBER

A CNC machine cannot cut a single panel without the tool. Good tires provide a smooth and safe ride. Similarly, **high quality, precision balanced tooling is needed to deliver the promise of productivity that comes with a CNC machine.**

THIS TOOLING GUIDE has been provided to help both newcomers and even the more experienced to the CNC machining industry. The purpose is to assist with the selection of cutting tools and clamping systems.

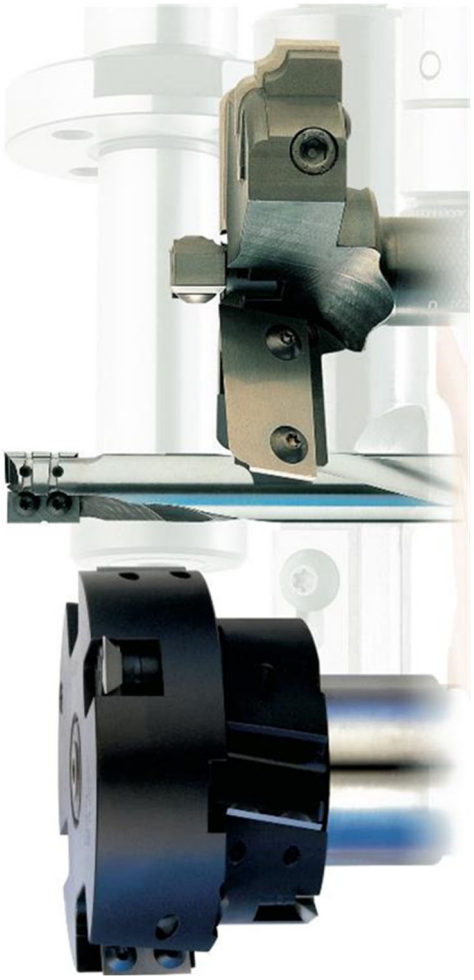
The ability to maximize tool life and achieve best possible surface finish is a prerequisite to obtaining the highest return on investment made in a CNC machine.

Its efficiency and productivity are only as good as the tooling paired with it, so understanding the most critical aspects of both tooling and accessories is essential.

GDP TOOLING

Introduction

CUTTING TOOL OPTIONS



This presentation will familiarize the CNC novice with the variety of tooling solutions that are available to meet specific cutting, profiling, drilling, grooving, and sawing tasks.

Furthermore, proper tool selection, material hold down, and dust extraction will increase the frequency of the following desirable CNC characteristics.

- Accuracy
- Repeatability
- Finish Quality
- Waste Reduction
- Spindle Bearings Upkeep
- Bottom Line Tooling Cost

Using tools within appropriate machining parameters, in particular, correct “chip load,” is important. **Chip load is the size/thickness of the chip being removed per flute/cutting edge with every revolution of the tool.** Going from a 2-flute bit to a 3-flute bit, the size of the chip is reduced by 33% if the feed rate or spindle speed (rpm) is not adjusted accordingly. A smaller chip will increase heat during the cut as the chips cannot be extracted fast enough and are re-cut into yet smaller particles. The resulting heat generated is very detrimental to tool life.

REMEMBER

Too big a chip load will decrease the finish.
Too small a chip will decrease tool life.

SAFETY FIRST

DIN Norm is the abbreviation for Deutsches Institut für Normung (often just referred to as Deutsche Industrienorm) is the German national institute for standardization of almost every field of technology. For the wood/composites industry, it has developed very extensive criteria and norms for the safe and effective operation of manufacturing machinery, cutting tools, clamping systems, and many other accessories. It's closest counterpart ANSI (American National Standards Institute), does not have a detailed norm for cutting tools and accessories.

When it comes to very specific tool design criteria with details such as minimum acceptable shank diameter based on the mass of the respective tool body being made, maximum knife protrusion beyond tool body (on insert tools) and countless other details that serve to protect both machine operators and the machine itself, in absence of such established norms in the U.S., we look to Germany's stringent DIN standards to dictate what we do.

While today's CNC machining centers are equipped with state-of-the-art features such as curtain guards, contact mats, guard fencing and more, a cutting tool spinning at 16,000+ rpm still poses a risk if sub-standard tools and clamping systems are employed and safety warnings are not followed.

Partner with someone who meets the standard!

Tool Holders and Clamping Systems

Clamping Systems Overview

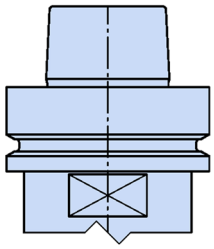
Drill Bit Adapter

IS0/SK30/BT/HSK-F & E Collet Chucks

Arbor Adapters for Cutters/ Blades with Bore

Saw Blade Arbor Adapter

MOST COMMON CLAMPING SYSTEM



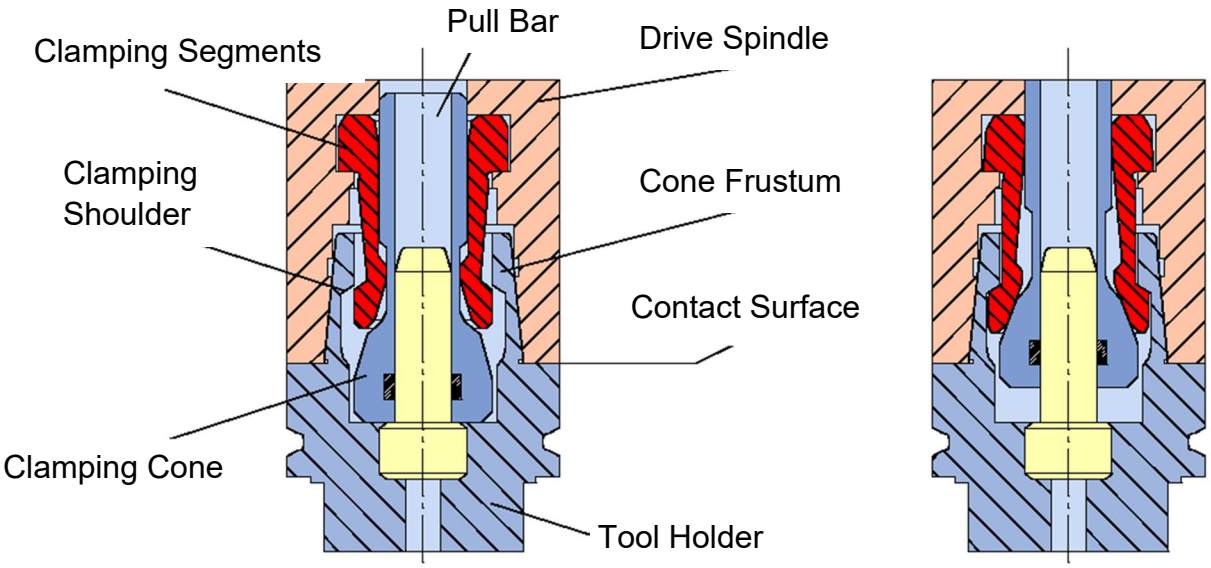
Developed in Germany, the HSK tool holder is the most widely used collet chuck on CNC routers and machining centers in the wood/composite industry as **it provides the highest degree of accuracy for high speed machining.**

It's important to purchase only the highest quality tool holders from reputable manufacturers to protect the machine spindle from potential damage.

As a precision machined interface between spindle and tool, it is balanced to G2.5 spec for 25,000 rpm. The HSK tool holder is manufactured in accordance with DIN 69893 and designed for use with automatic tool changers.

HSK (*Hohl-Schaft-Kegel*), translated from the German language, means "hollow shank taper" and describes the part of an HSK tool holder that connects to the machine. This taper is machined to the highest precision both radially and axially, of .0001" per DIN Norm.

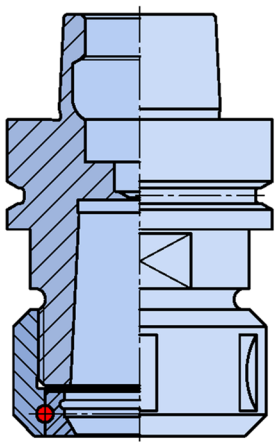
HSK INTERFACE ANATOMY (DIN 69 893)



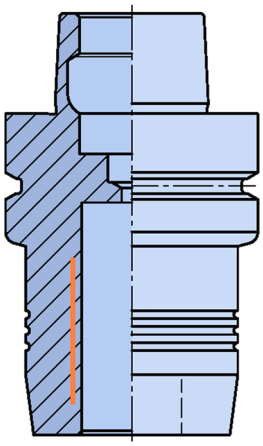
Router Tool Holders

MOST COMMON HSK-F 63 TOOL HOLDER STYLES

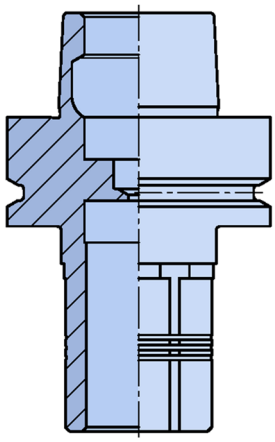
On any machine that has the HSK-F 63 tool interface, the tool holder styles are interchangeable



Standard Collet Chuck
Preziso Collet Chuck



Hydro Chuck



Heat Shrink Chuck

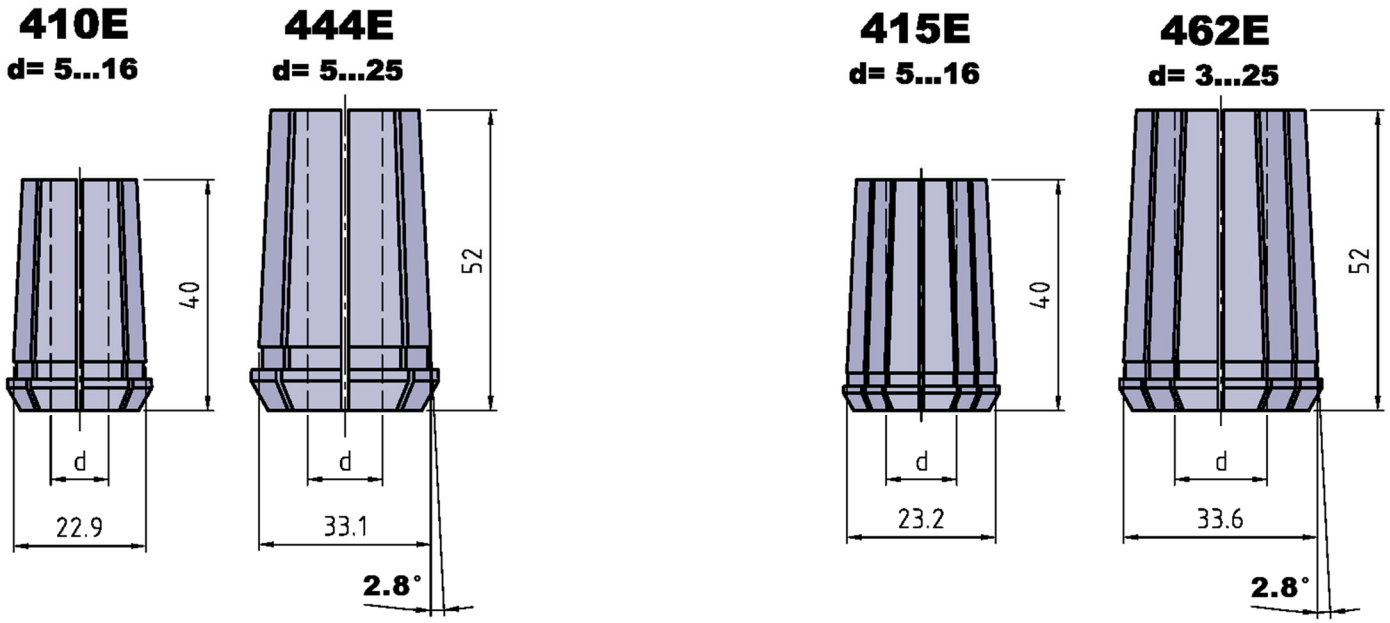
ACCURACY OF THE DIFFERENT TOOL HOLDER STYLES

Standard Chuck	Preziso® Chuck	Hydro Chuck	Heat Shrink Chuck
.006mm-.010mm*	.003mm	.003mm	.003mm
			
Comes in 3 styles using either ER32, ER40 or RDO/ SYOZ collets	Uses special RDO35 high precision collets	No collet	No collet

*depends on collet style used

Collet Style & Range Tolerance

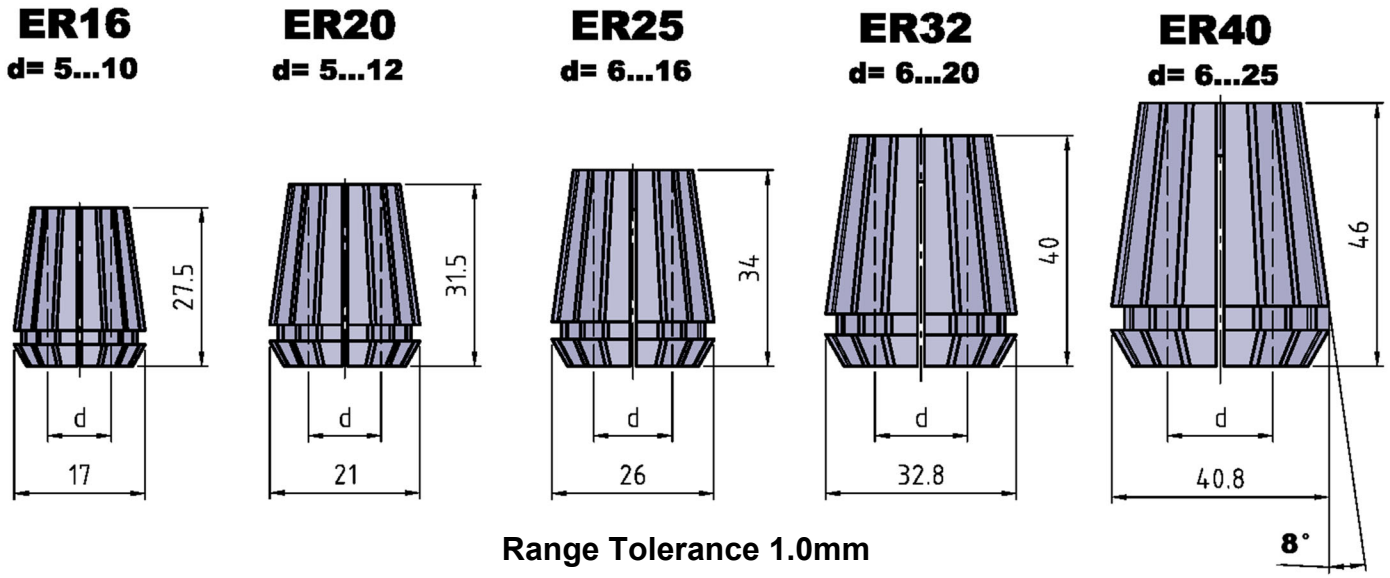
RDO/SYOZ STYLE



Range Tolerance 0.15mm
(High Precision Version)

Range Tolerance 0.5mm
(Standard Version)

ER STYLE



Range Tolerance 1.0mm

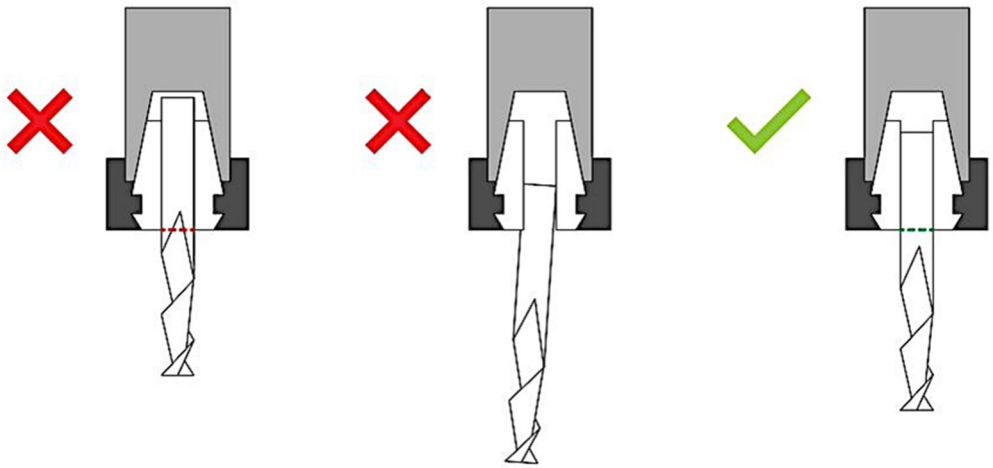
GDP TOOLING

Collet Style & Range Tolerance

COLLETS—NOTEWORTHY INFORMATION

Collets wear out! Over time, the spring steel loses elasticity due to wear and heat. Collets that are not replaced will eventually fail. Collets are inexpensive, and **failure to replace them regularly will increase tool cost due to unnecessary vibration and runout, tool chattering and breakage.** They can do considerable damage to a spindle, so adhering to a collet maintenance schedule is preferable to costly repairs. Below are some helpful tips:

- Clean collets and collet nuts when changing tools!
- Use proper torque! Investment in a torque wrench to tighten the collet nut is a prerequisite to extending tool life and reducing breakage!
- Pay attention to clamping the tool correctly! (see below) With proper tool setup, the collet should be filled to 80% with the tool shank. Shorter will allow deflection.

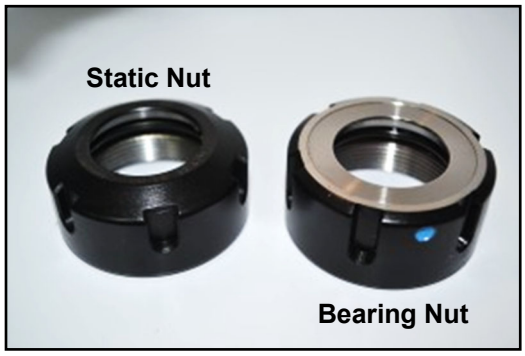


Replace collets every 500 machine hours for optimum tool life! Prevent costly spindle repairs!

COLLET NUTS OVERVIEW

The collet nut, and accuracy thereof, is an important component of collet tool holder setup. **A high precision collet nut will have a ball bearing located at its base which reduces the overall runout and improves clamping force.** It also prevents tool slippage. The static nut does not clamp as accurately as the bearing version (see photos on next page).

Collet Nuts



The bearing in the collet nut also facilitates the ability to run either CW or CCW tools in the same tool holder. This is not possible with a static nut, and a lefthand toolholder would be needed.

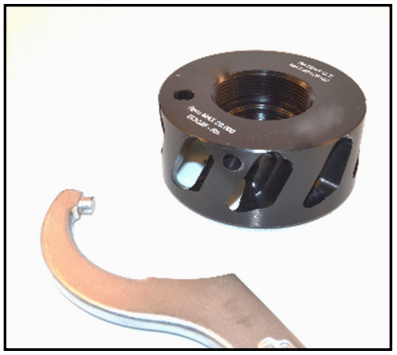
WHICH STYLE TOOL HOLDER TO USE

For routine machining of wood and composite materials, the standard collet chuck style tool holder is an excellent choice. It has an accuracy of .006-0.01mm depending on which collet style is used. For the more discriminate user and critical machining applications, the option of a Preziso® Precision Chuck, Hydro Chuck, or Heat Shrink chuck are good options.

Standard Chuck	Preziso® Chuck
• Excellent general use choice • This version is supplied with most new machines and adequate for general machining tasks	• Excellent choice to reduce vibration when cutting very hard materials • Improved efficiencies over standard tool holders, optimizes tool life • Good choice to increase tool life with solid carbide spiral bits • No additional equipment needed
Hydro Chuck	**Heat Shrink Chuck**
• Rigid tool stability • Extremely high clamping force • Saves expense of replacing collets regularly • Initial higher cost	• Best for the Spoil Board (Fly) Cutter, all Insert Tools, and Diamond Tools • Improved efficiencies over standard tool holders, increases tool life, and improves finish • Not suitable for solid carbide tools and tools requiring frequent change since an additional investment in a heat induction unit is required • Saves expense of replacing collets regularly • For best results, we recommend ordering insert tools and diamond tools pre-mounted on heat shrink chucks

Cyclone Dust Nut

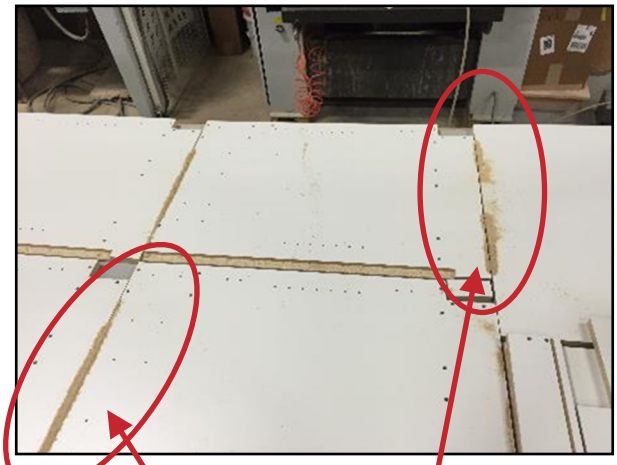
CYCLONE DUST NUT OVERVIEW



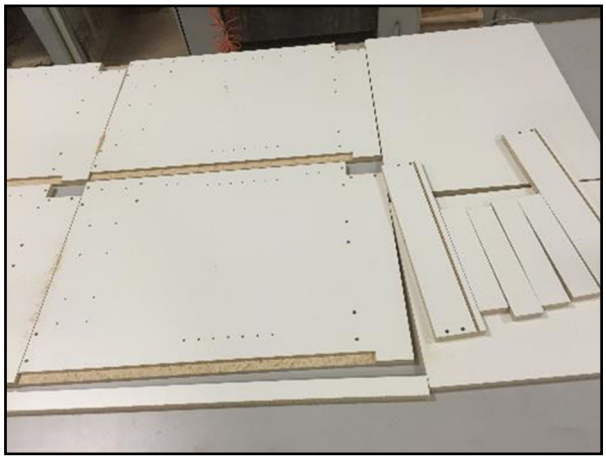
A dust nut can be especially useful if too much dust is remaining on the table. More importantly, it's a safer and healthier work environment to reduce airborne exposure to dust particles. **The Cyclone Dust Nut takes the place of the regular bearing collet nut on the tool holder.**

Through its geometry, it creates a whirlwind effect that sucks the chips out of the cut and into the dust extraction system. The Cyclone must be kept close to the shroud/table to function properly, so it's best to have the shortest possible cut length on the router bit to allow the nut to be positioned as close to the table as possible.

Our company offers a demo/test nut to try so customer knows if it will work in their application before buying.



Note where dust has collected

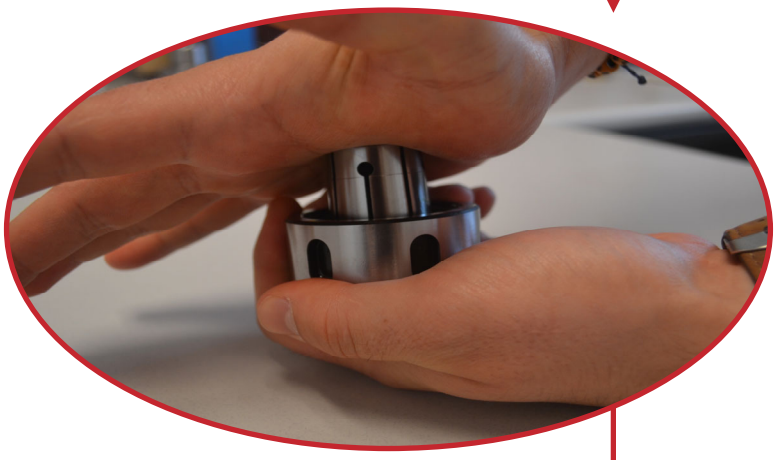


Vastly reduced dust with the cyclone dust nut!

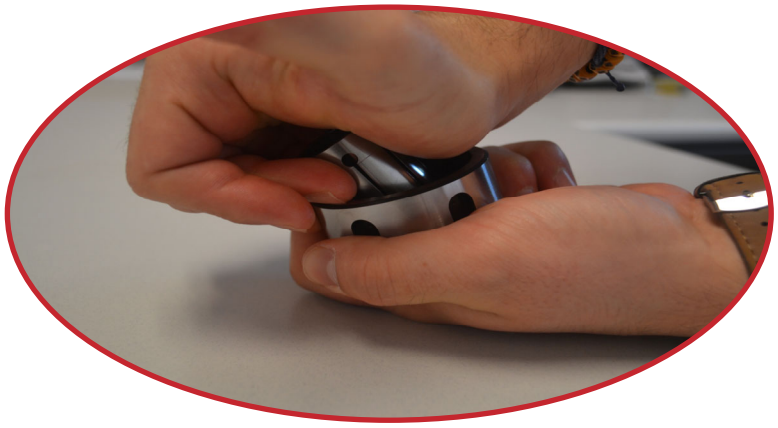
Collet Installation Best Practices

COLLET REPLACEMENT

First, click the collet into position by pressing only one side



Then, push down on the other side to lock it

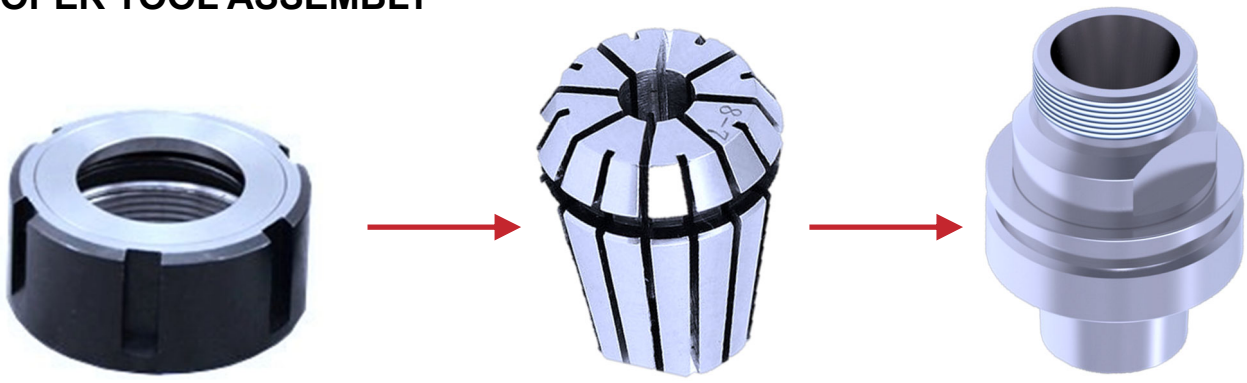


To release the collet, use the ball of your hand to push sideways

GDP TOOLING

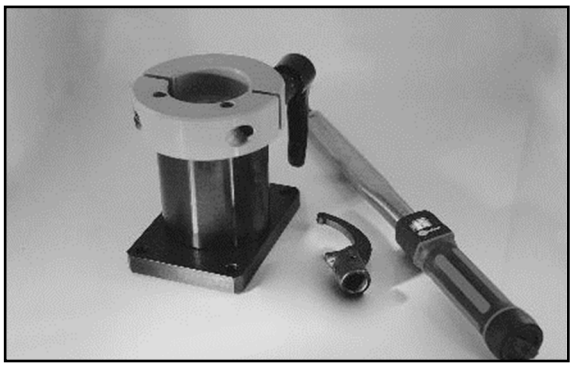
Collet Installation Best Practices

PROPER TOOL ASSEMBLY



1. Snap the collet into the collet nut or cyclone nut.
2. Place the tool into the collet paying close attention to the proper tool depth in the collet. Tighten the collet nut manually onto the tool holder.
3. Position and secure the tool holder in a tool setup fixture as shown below.
4. Tighten the collet nut with a torque wrench to the **correct torque** based on the collet style used.

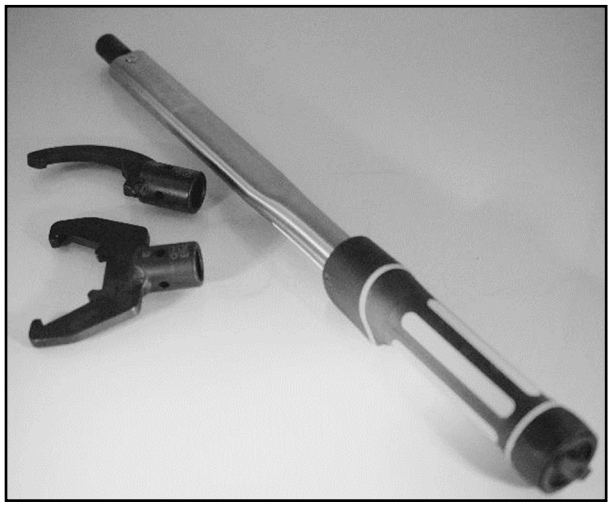
Collet Style	Recommended Torque
ER 20	59 ft/lbs
ER 32	100 ft/lbs
ER 40	130 ft/lbs
RDO 35 (SYOZ25)	90 ft/lbs

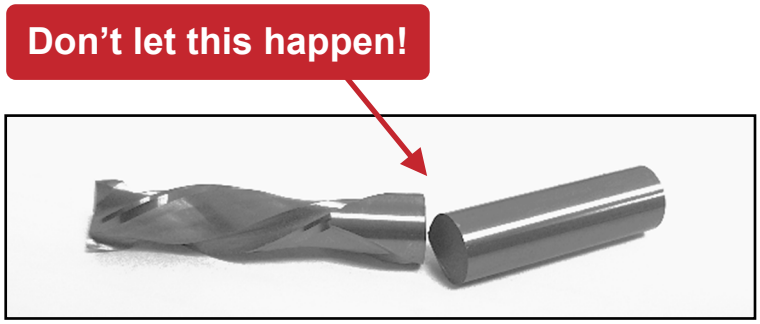


Setup Fixture (no gauge) and torque wrench

Setup Fixture with gauge

PROPER TORQUE

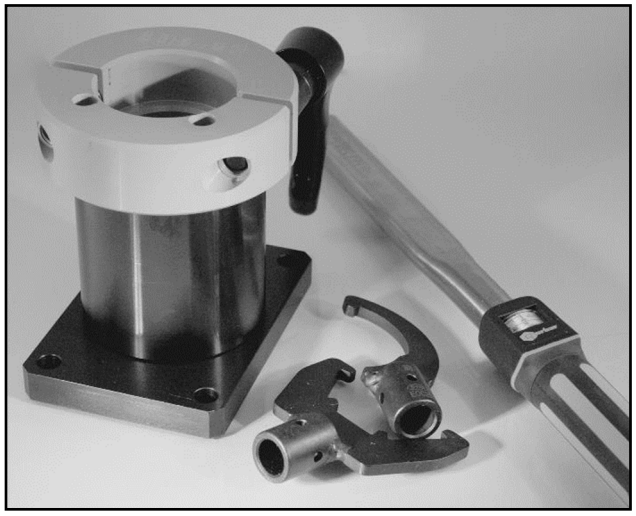




Typical breakage pattern resulting from improper torque!

Incorrect torque when tightening the collet nut will result in poor cutting performance, premature collet wear, tool slippage during the cutting process, and over-torquing can often result in tool breakage and chattering.

Investment in a torque wrench has paid for itself after the first few bits broken due to over torquing. **Use the torque wrench only for tightening the collet nut. Use a standard wrench to remove it. Otherwise, the torque wrench can lose calibration accuracy.**



TOOL SETUP FIXTURE

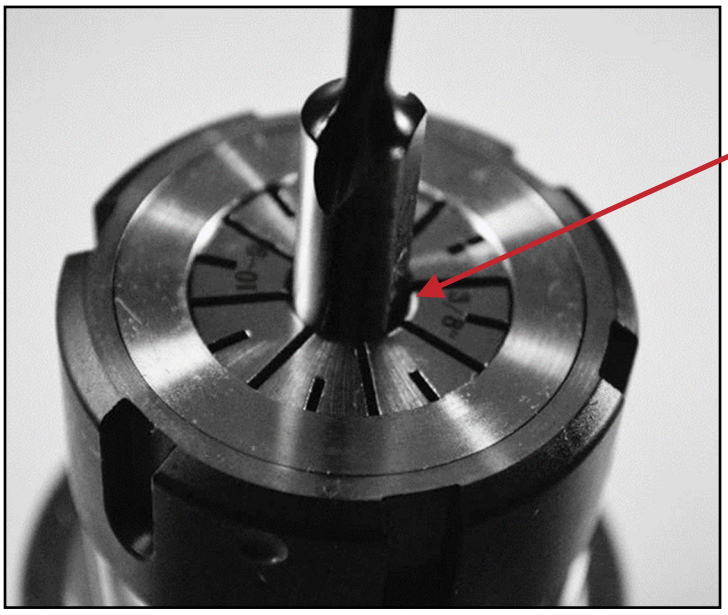
A setup fixture and torque wrench are a necessary investment to achieve optimum tool life. These avoid tool breakage, inferior finish, premature wear, and a high tooling cost. After spending over six figures on a CNC machine, this is not the time to pinch pennies.

Drill Bits and Adapters

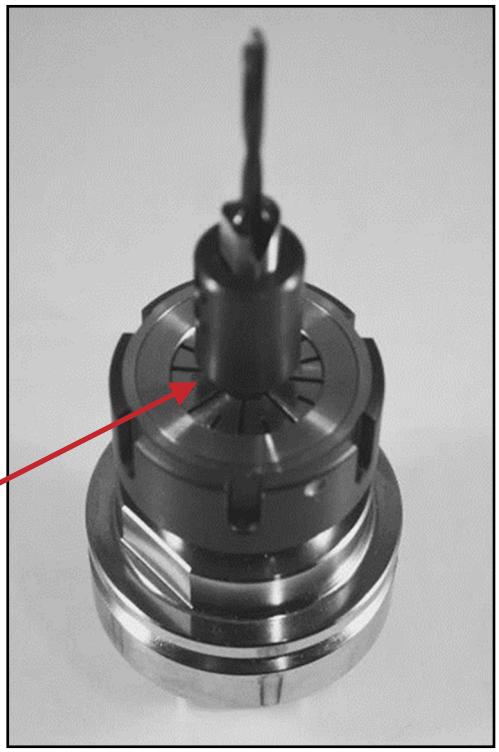
Using Drill Bits on a CNC

PROPER TOOL ASSEMBLY

Using typical 10mm shank drill bits with a set-screw flat area on the shank in a collet-style tool holder can result in oval holes. The drill bit shank cannot be clamped in the collet evenly around its circumference due to the flat on the tool shank present for set screw tightening. The gap that remains between the collet wall and tool shank in the flat area can cause the bit to deflect, possibly wobble, and not provide a clean hole. The solution is to get a small and inexpensive drill bit adapter that has a 10mm cylindrical shank (without flat) to go into the collet, and the female end of the adapter accepts the 10mm drill shank with flat. It is tightened in with a set screw. For CNC use, the 70mm drill bit length should be chosen over the 57mm length..

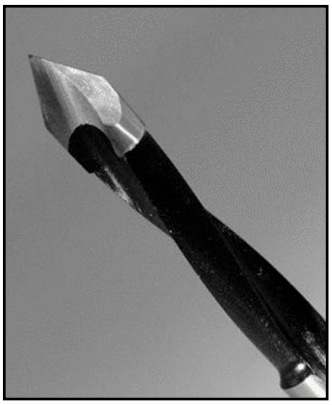


Gap

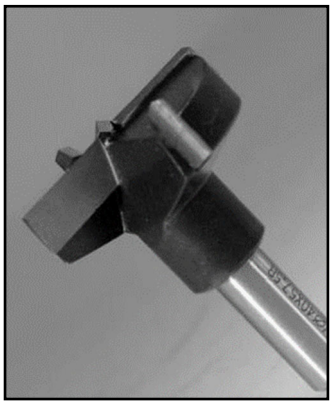


Drill Adapter

DRILL BIT SELECTION



For holes that will go through the material, use **V-Point Bits** (Thru-Hole Bits) that prevent surface tear-out on the bottom of the cut.



For hinge pockets, the **hinge boring bit** is the best option. The center point and outside spurs provide a clean edge hole and prevent the bit from “walking.”

For blind holes, a **brad point/dowel bit** is required. It has a center point and outside spurs to cut a clean hole. For pilot holes, this bit is available in solid carbide in 1/8” diameter.

CNC DRILL ADAPTER

This **Universal Drill Adapter** is the best and only option if drilling many different cylindrical drill bit sizes for which collets are not always available.

Recommended RPM	Speed for Drilling on CNC Feedrate (m/min)
3,000	1-2
4,500	1.5-2.5
6,000	2-3.5
9,000	2.5-5.5

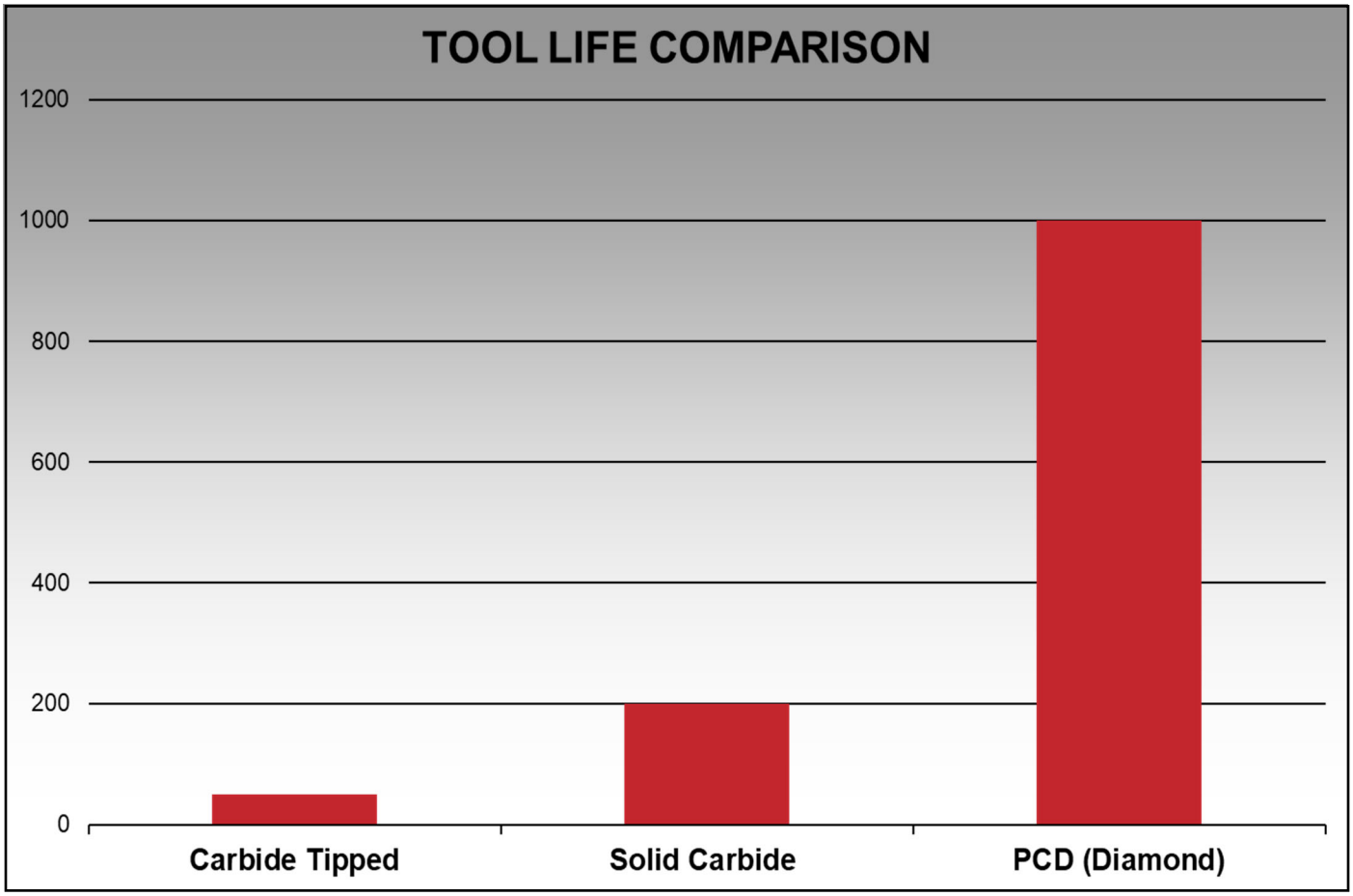
RPM and Feed Rate: When drilling or boring holes on a CNC machine, your spindle speed and feed rate must be adjusted to within the proper parameters for drill/boring bits.

GDP TOOLING

Sizing, Jointing, Rebates, Dado Cuts and Grooving

Router Bits

STRAIGHT ROUTER BITS—TOOL MATERIALS

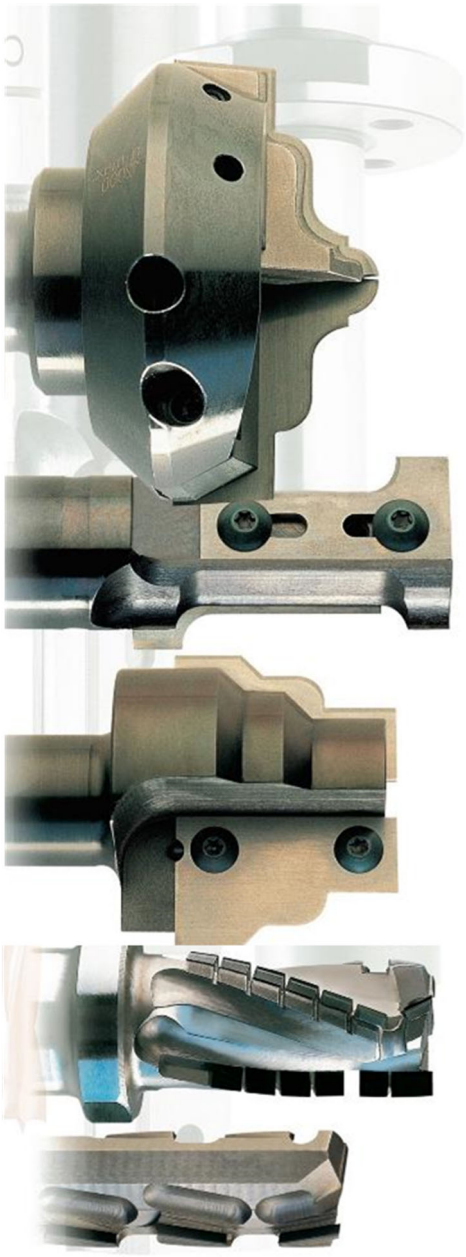


The best options for CNC machining are either solid carbide bit or PCD (polycrystalline diamond).

PCD tooling achieves the best return on investment when it is dedicated to cutting homogeneous material and not interchanged between materials (e.g. composite and wood). **Use dedicated tools per material!**

The only time a carbide-tipped tool is an economical solution for use on a CNC is for cutting a small profiled proto-type (i.e. one-off).

ROUTER BITS OVERVIEW



Depending on product and volume being machined, the selection of the most suitable cutting tool requires a bit of homework if best performance and finish relative to investment is desired.

In the following pages, we present the most common and cost effective router bit options based on application.

For **straight cuts** such as sizing, jointing, rebating, grooving and dados, the options available are solid carbide, carbide insert, or PCD (polycrystalline diamond).

For **profile routing applications**, options are either carbide insert or PCD, and for proto-type or “one-off” needs, a knife adapter provides a cost-effective solution.

In addition to the many standard tool designs, **it is often necessary to customize a tool for a particular requirement in order to provide a more economical option over time.**

Many tools styles can effortlessly produce the same cut, so researching cost, time and yield based on tool choice can have a substantial impact on a tooling budget.

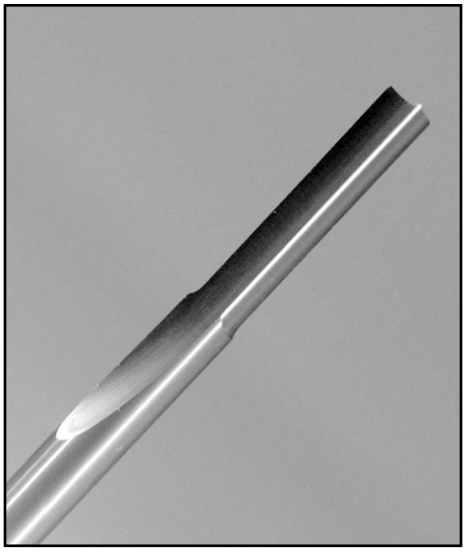
Solid Carbide Bits

OVERVIEW

Solid carbide router bits come in a variety of tool geometries, number of flutes, and edge grind. Some of the most popular ones are listed below and described in more detail on the following pages.

- O-Flute, Straight (Soft Wood/Soft Plastic)
- O-Flute, Up Cut (Plastics, Aluminum)
- Up Cut
- Ballnose Up Cut
- Down Cut
- Compression (Mortise Compression)
- Compression Chip Breaker
- Up Cut Chip Breaker
- Down Cut Chip Breaker
- Up Cut Rougher
- Down Cut Rougher
- Single, Two, and Three Flutes

REMEMBER: More flutes does not equal better finish! More flutes means you have faster feed rate capability! Refer to the chip load information on pages 40-41 to calculate your needs.



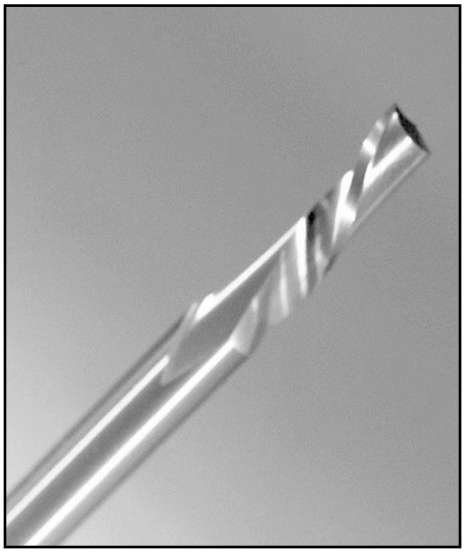
O-FLUTE BIT

This image shows a **solid carbide, straight O-Flute**. The flute shape is ground into what looks like a half-circle.

This particular geometry is used for cutting flexible plastics. The flute is straight and helps keep a light weight flexible plastic from moving or lifting on the router table during machining.

Use this tool on materials such as:

- Polycarbonate
- ABS
- Polystyrene
- PVC
- Other flexible plastics



O-FLUTE UP CUT BIT

The **O-Flute up cut bit** features the same O-Flute geometry needed for efficient chip removal when routing plastic, and it also has an upward shearing angle to bring the chips out of the cut and provide cleanest possible edge finish. This tool is the preferred router bit for hard and rigid plastic materials.

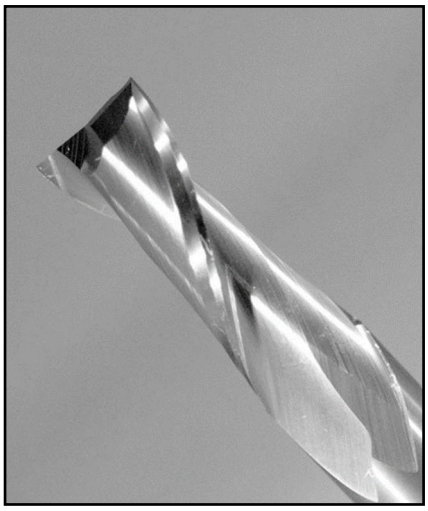
Use this tool on materials such as:

- Acrylics
- Nylons
- Plexiglass
- Other rigid plastics

Solid Carbide Bits

UP CUT SPIRAL

The **up cut spiral bit** is used when getting the chips out of the cut is critically important, or when the material is laminated/ coated on one side only and is being machined face down. This tool geometry does present a risk that the top of the material being cut can chip or fray.



This tool is a good choice for mortise and tenon cutting needs and also provides an excellent option for short runs on Corian or phenolic (longer runs should consider a PCD diamond bit as it is more cost-effective).

Up cut bits provide the ability to feed faster than down cut bits as the chips are pulled out of the cut by the upward shearing action and keep the tool running cooler.

BALLNOSE UP CUT BIT

A **ballnose tool** is a great choice for cove and fluting operations, but it is also the tool of choice when complex shapes have to be surfaced. A flat bottom bit would leave transition lines and poor finish, but a ball nose bit provides seamless passes due to its rounded shape.

Another option for this tool is a carbide insert version (pictured below), which is always much more cost effective if the project is ongoing with a continuous tooling need.

Carbide Ballnose Up Cut Insert

DOWN CUT BIT

The **down cut spiral bit** provides a superb, top surface finish, but it does run the risk (depending on the application) of pushing the chips into the cut and bogging down the bit. With good dust extraction and proper chip load, this should not be a problem. It is often selected for doing grooves, dados, and rebate cuts.

If there is an ongoing dado or rebate cut requirement, an insert bit will decrease cost considerably within a short period of time if the tool is available in the diameter needed.

Down Cut Insert Bit

REMEMBER: always use the shortest possible cut length for dados, grooves, and rebates—longer tools will have more deflection and can break more easily or provide poor finish due to deflection when cut length is excessive.

Solid Carbide Bits

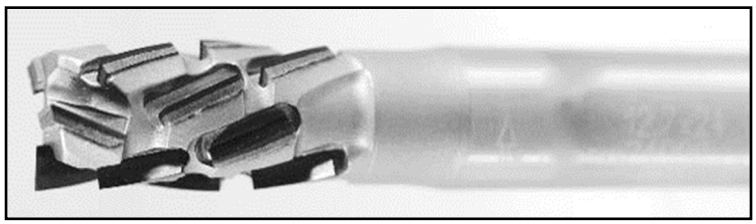
COMPRESSION BIT

The **compression bit** is designed to cut materials that have a laminate, melamine, HPL, paint, paper etc. on both top and bottom surface. The cutting flutes have opposite shear angle geometry and cut toward the center of the material, thus providing clean surface top and bottom.

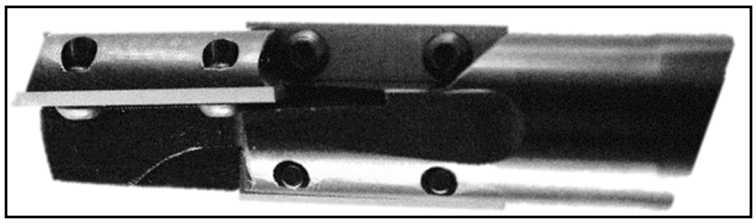
This is the most popular style bit used in composite panel processing and is available in many different executions. They range from a variety of carbide grades defined for wood, composite material and melamine, and also coated versions.

If using a 3/4" or larger compression bit, an insert or a PCD diamond tool offer the lower cost option for any ongoing need.

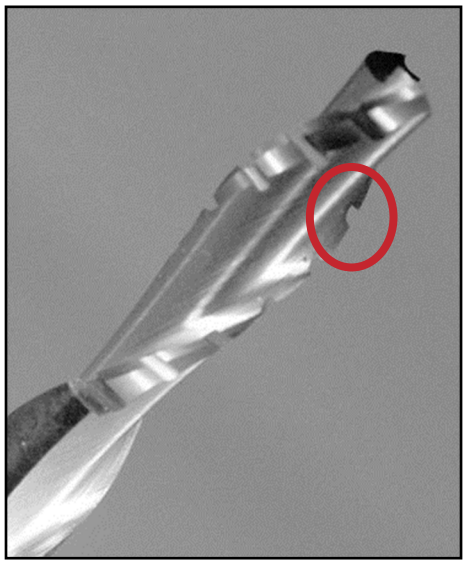
These bits come in multiple carbide grades. Pick the right one for the job!



PCD Compression Bit



Insert Bit for >3/4" Diameter



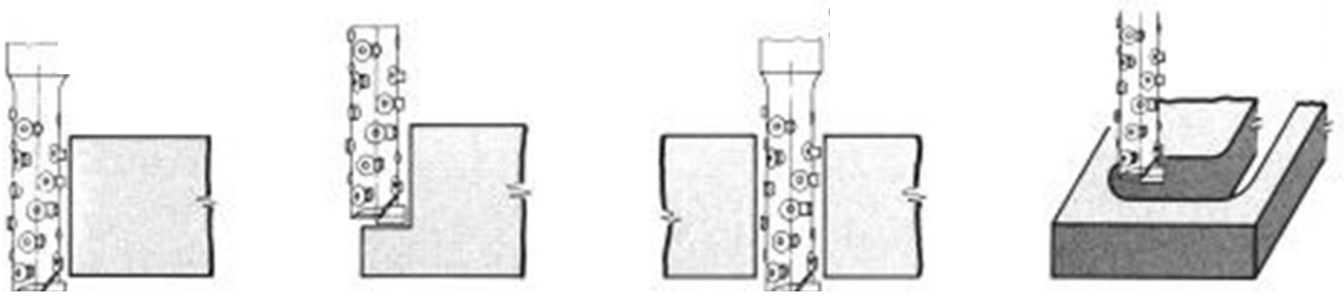
COMPRESSION CHIP BREAKER BIT

The **compression chip breaker bit** is the best choice for cutting plywood and OSB panels where a good surface finish on both sides is expected. The chip breakers, which are notches machined in an offset pattern into the flutes, facilitate faster feed rate and breaking up the larger chips created by these materials. While the chip breaker is essentially more of a hogging tool, it does provide excellent machining characteristics as well as good finish in the compression style.

To the left is an insert tool option for high feed speed requirements on plywood panels and other composite materials.

UPCUT OR DOWNCUT CHIPBREAKER BIT

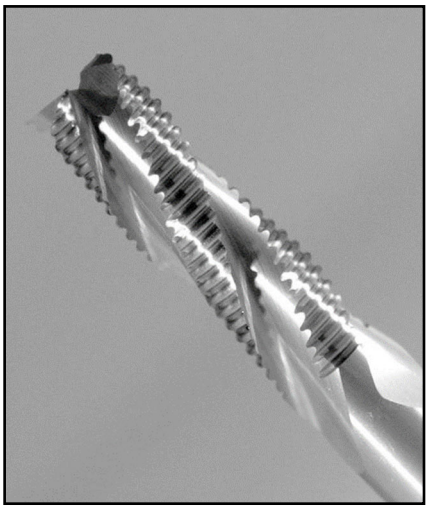
The **up or down cut chip breaker** is a great choice for cutting composite panels and plywood where a good surface finish on one sided is needed (down cut for best finish on top, up cut for matching face-down). The insert option pictured below features a selection of insert knives that are either straight, up or down shear, or even compression style and can be arranged as needed on the tool body as shown below. This allows them to be positioned for best possible finish results.



Solid Carbide Bits

UP CUT OR DOWN CUT ROUGHING BIT

A **roughing bit** is designed to remove a lot of material quickly, but as the name implies, it does leave a rough edge which must be cleaned up with a finishing pass using another tool.



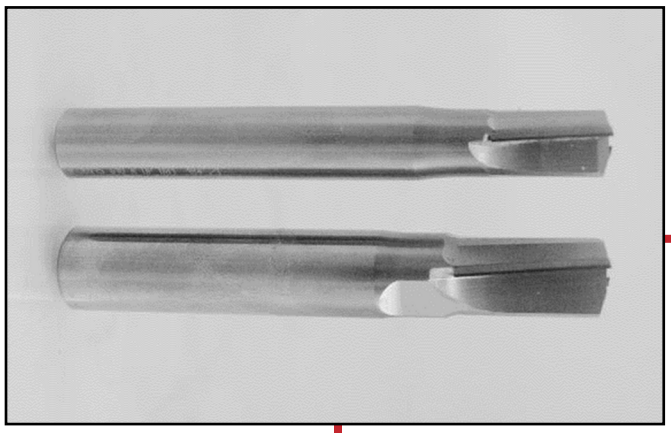
Typical feed rate for these 3-flute tools is 800"/min and up.

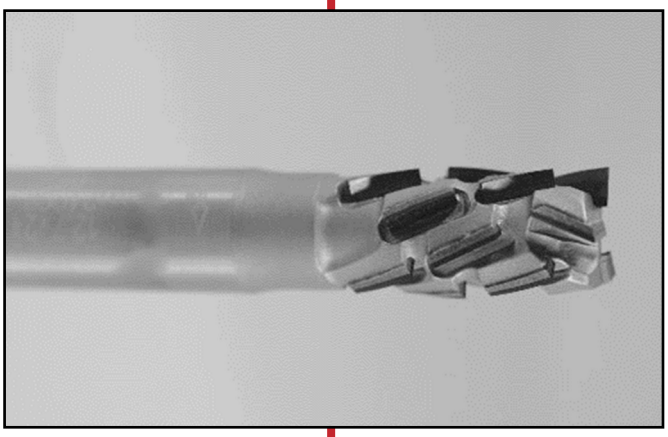
A very cost effective option is an insert roughing tool (pictured above) which is available in either carbide or diamond (PCD) inserts and provides a tremendous cost saving for high volume operations.

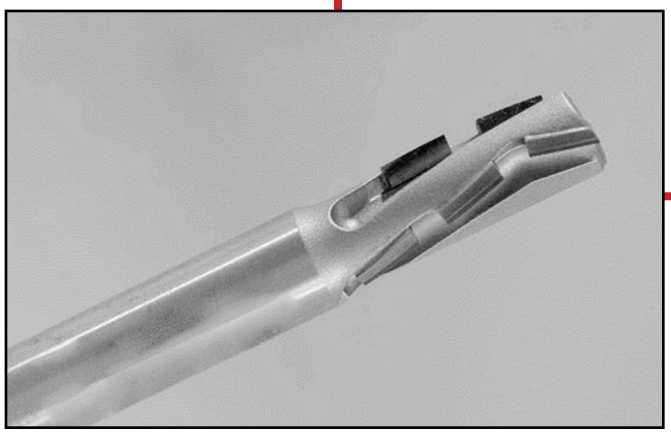
PCD/Diamond Bits

DIAMOND TIPPED ROUTER BITS

Whether diamond tooling should be a consideration for a particular application is discussed in the next few pages. **Diamond router bits** come in a variety of styles such as high shear, single flute, 2-flute or 3-flute disposable, single segment opposite shear, etc.







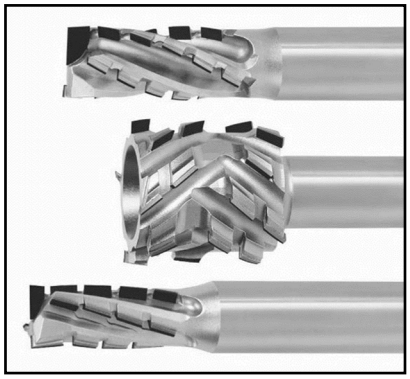
Diamond tools are not all equal. Some are designed to be disposable, others can be sharpened one or more times. These factors should be considered when shopping for PCD tooling.

PCD/Diamond Bits

NESTING TIPS TO REMEMBER

- Always adapt the cutting length of the router tool to the panel thickness (i.e. cutting length should be minimally longer than the panel thickness).
- Always choose the stronger tool (i.e. cut length not too long and diameter not too small).
- Material hold-down must fit the tool. That means, select small diameter tools for parts that are prone to moving.
- Chip clearance is better on larger diameter tools; therefore, when running high feed rates and thicker panels, use a diameter of 5/8" or greater.
- Select highest accuracy clamping tools (Heat Shrink, Hydro Chuck, or Preziso)
- If using collet chucks, replace collet regularly (every 500 machine hours!)

DECIDING BETWEEN CARBIDE OR DIAMOND TOOLS



There is no debate that PCD (polycrystalline diamond) will outperform solid carbide tooling in wood and composite material by many multiples and be the more economical choice over time. Until recently, there was a good argument to stay away from diamond router bits since most companies promoting them only offered single flute on the diameters of 1/2" or less. This did not facilitate high feed rates. That has been changed with the introduction of 3/8" and 1/2" PCD bits that are 2-flute!

If you want to achieve the best possible tool life and save substantial amounts of money in the process, you would be remiss to not investigate the cost of running a diamond tool for your biggest projects! Whether sizing or profiling, diamond tooling should always be used on homogenous material and same panel thickness to maximize the tool life and performance. Most diamond tools can be sharpened a few times and will last longest when paired with a heat shrink tool holder for highest accuracy.

COST COMPARISON OF A CARBIDE COMPRESSION VS DIAMOND TOOL

The below comparison is based on the carbide and diamond tools both being sharpened twice, but it does not include setup cost or machine downtime. It also does not factor in that after sharpening a carbide spiral, clearance is lost and thus lower tool life cycle is achieved.

Life Cycle estimate 24:1—a conservative value!

FORMULA

Cost per panel = $\frac{\text{Tool Cost (number of sharpenings x sharpenings cost/each)}}{\text{1 + number of re-sharpenings x tool life (panel count)}}$

Carbide Tool = $\frac{\$65 + (2 \times \$16.50)}{(1+2) \times 85}$ = $\frac{\$98.00}{255}$ = **$0.38 per panel**

Diamond Tool = $\frac{\$388 + (2 \times \$125.00)}{(1+2) \times 3000}$ = $\frac{\$638.00}{6{,}000}$ = **$0.11 per panel**

When Not to Use Diamond Tools

- When materials and material thickness varies and having a diamond tool for each operation is not feasible.
- If initial cost outlay is critical and a higher upfront tool price isn't tolerated for cash flow reasons (even though it will save much over time).
- If material is prone to inclusion of staples, nails, etc. (diamond tools are susceptible to impact damage)
- If the machine is older and has excessive spindle runout, or parts tend to move on the table (carbide is the better choice)
- If machine operators are poorly disciplined and wouldn't handle a diamond tool with the gentle respect it deserves.
- If customer is not prepared to invest in a complete setup to ensure the diamond tool is capable of performing to expectations (i.e. doesn't want to replace the collet or invest in a heat shrink tool holder)
- Customer is cutting parts that require plunging straight down into the material (this creates a lot of heat that will cause the PCD tool to wear prematurely)

IMPORTANT–Chip Loads

The next two pages contain information that is CRITICALLY IMPORTANT to preventing tool breakage and maximizing tool life.

Please take the time to review.

FINDING YOUR OPTIMUM CHIP LOAD

When the chip is too small, the cutting action will generate heat in the cut and will cause the cutting edges of the tool to deteriorate prematurely. A larger generated chip within the reference range for a particular material will achieve the longest tool life.

You can find your **optimum chip load**, which will allow you to maximize productivity and get the best tool life and lowest cost per panel, if you follow the steps below:

1. Start by using the recommended chip load and slowly increase your feed rate until the finish quality becomes unacceptable.
2. Slowly decrease feed rate again until desired finish is restored. Make note of your feed rate.
3. Decrease the machine RPMs until the finish deteriorates.
4. Once that occurs, increase RPMs until finish is once again restored. At this point, you have found the “sweet spot.”

RPM/FEED SPEED

Correct chip load is an important factor to extend tool life and avoid premature tool wear. **The chip load is the size of the chip the tool makes during the cutting cycle** and it is calculated based on the number of flutes on the tool, the spindle speed and the feed progression. The below short is a starting point reference range only*.

Tool Diameter	Hardwood	Plywood	MDF/ Particleboard	Soft Plastic	Hard Plastic	Acrylic	Solid Surface
1/8"	.003" - .005"	.004" - .006"	.004" - .007"	.003" - .006"	.002" - .004"	.003" - .005"	.002" - .004"
1/4"	.009" - .012"	.011" - .013"	.013" - .016"	.007" - .010"	.006" - .008"	.008" - .009"	.006" - .009"
3/8"	.014" - .018"	.017" - .020"	.020" - .023"	.010" - .012"	.008" - .010"	.010" - .012"	.008" - .010"
1/2"	.019" - .021"	.021" - .023"	.025" - .027"	.012" - .016"	.010" - .012"	.012" - .015"	.010" - .012"

***The above range is based on cutting depth being equal to cutting diameter. For deeper cuts, adjust the chip load as follows: for 2x diameter, reduce by 25%; 3x diameter reduce by 50%.**

CALCULATING CHIP LOAD

Calculator available here: www.gdptooling.com/chipload-calc

NOTE: If your machine displays feed rate in metric values, take the metric value (e.g. 8 meters), multiply by 39.37 to obtain the inch equivalent for your calculation.

Chip Load Formula

$$\text{Chip Load} = \frac{\text{Feed Rate (inches per min)}}{\text{(RPM x number of flutes)}}$$

$$\text{Example} = \frac{\text{Feed rate of 500" per min}}{\text{(16,000 rpm x 2 flutes)}} = \frac{500}{32{,}000} = \textbf{Chip Load is 0.15}$$

To Calculate Feed Rate on the Fly

Example: Using a 2-flute, 3/8" compression bit cutting into MDF with an RPM of 16,000, your formula is: (16,000x2) x 0.23 (chip load per schedule) = 736"/minute (adjust based on cut depth).

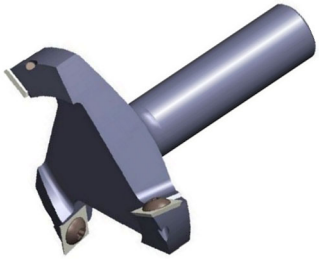
Other Common CNC Tooling Solutions

Other Common Tooling Solutions

SPOIL BOARD/FLY CUTTER

A clean and level spoil board is prerequisite to good vacuum hold-down and machining accuracy.

When setting up machining program, the routing tool should extend beyond the work piece and penetrate the spoil board by 0.3mm - 0.6mm. Periodic resurfacing of the spoil board to maintain a flat, even surface is most efficiently performed with a large diameter fly cutter. They are available in 40mm, 80mm, or 100mm diameter.

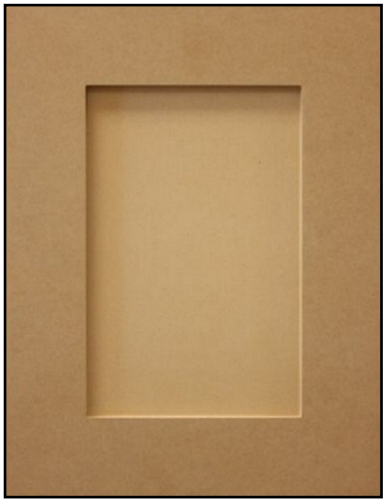


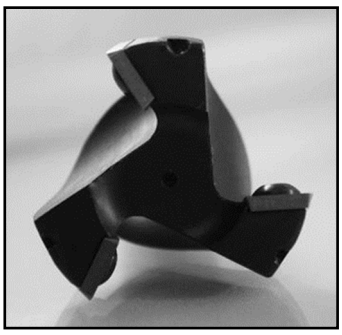
For best performance, these insert cutters should all be mounted in an HSK heat shrink tool holder.

HIGH QUALITY SURFACE PLANING TOOL

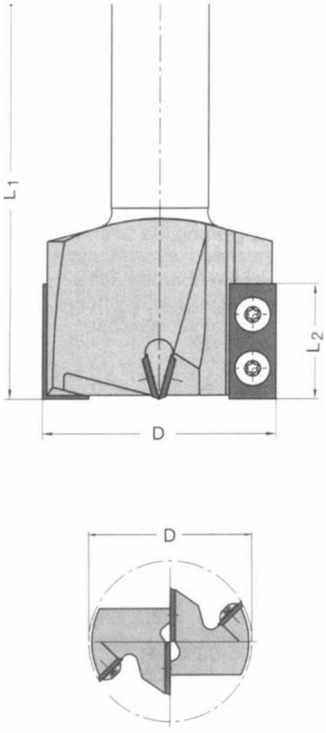
An insert tool with special knives and edge radius provides a superb surface finish for applications such as MDF shaker style cabinet door cut outs. Using two additional tools (a 3/8" solid carbide bit and a 1/16" solid carbide bit) provide the ability to produce a square corner cut out in MDF doors. For optimum results, this tool should be used on HSK heat shrink tool holder.

Sample Product Use





Insert knives also available in PCD (diamond)

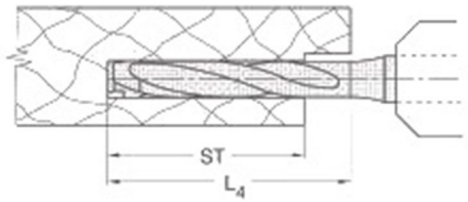


INSERT REBATE CUTTER

For rebate or deeper surface planing cuts, these insert tools provide a very economical solution as the replacement insert knives (when the tool is dull) usually cost under $6/cycle. The solid carbide inserts are available in different carbide grades to facilitate efficient cuts in either wood or composites.

DEEP POCKET MORTISE CUTS

The further the cutting edge of a tool gets away from its clamping source, the more deflection is created, and the bigger the risk of tool breakage. Considering the rule of thumb that cut length shouldn't exceed 3x the cut diameter, cutting a deep mortise pocket can represent a challenge. **Special, deep pocket mortise bits** solve that problem as they are made from a very special, high density alloy that prevents deflection and tool breakage despite the depth of cut.



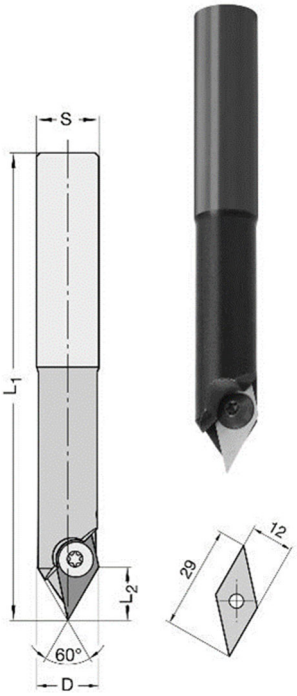
Recommended RPM: 12,000 -18,000
Gradual Plunge Feed: 12m/minute (up/down)
Max Cut Progression: 8mm-10mm solid wood, 15mm composites

GDP TOOLING

Other Common Tooling Solutions

V-GROOVE AND ANGLE CUTS

For mitre folds, insert v-groove bits are available as standards for 45, 60, and 90 degree (included angle) cuts. Other angles can be produced as custom tools. The insert knives are double sided and provide two life cycles each.



Insert v-grove bits are also a great solution for lettering, engraving, and decorative cuts as well as beveling the inside edges of shaker doors and square corner cut outs.

TOOLING FOR SPECIFIC MATERIALS

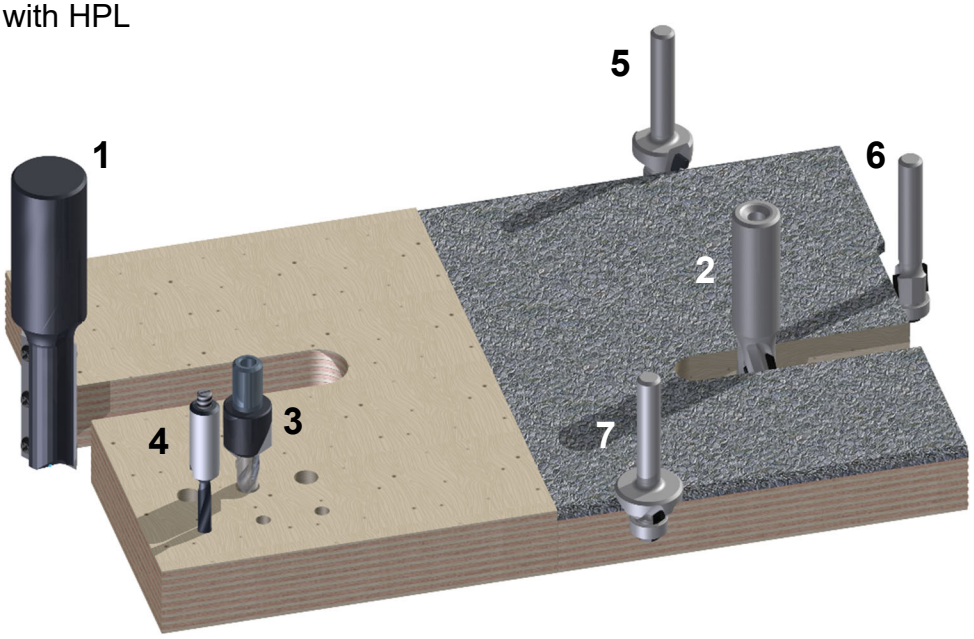
A solid carbide tool is often selected for most cutting challenges on a CNC machine. However, this is definitely not always the most cost effective solution.

As you will see in the following pages, there are many options to choose from. Therefore, we recommend to look for the most economical option for the job at hand that doesn't compromise end result.

Carbide insert tools or diamond tools offer a lower cost alternative. These options are often overlooked by short-sighted decisions based on initial cost—which can be deceptive. In the following pages, we present some examples of slotting, grooving, edging, drilling, and profiling in a variety of materials.

Machining Multiplex, Veneered Composite Panels

Similar Applications: Plywood and layered wood panels of different wood species, laminated with HPL

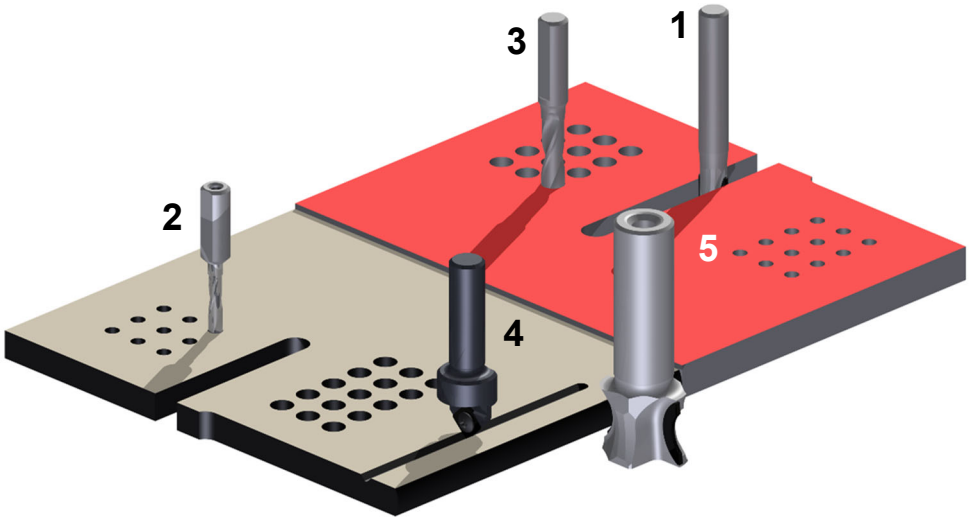


Legend

1. Insert Bit (22025)
2. 3-Flute PCD Bit (13750)
3. Solid Carbide Through Hole Bit (6028SC)
4. Carbide-tipped Dowel Bit
5. PCD Round Over Bit
6. PCD Jointing Bit
7. PCD Bevel

Machining Phenolic (Left) Cement Fiber Board (Right)

Similar Applications: Cement Fiber containing walls, Swiss Pearl, and particleboard panels with cement fiber content



Legend

1. PCD 6815 2-Flute Bit
2. PCD Dowel Drill Bit
3. PCD Dowel Drill Bit
4. PCD Engraving/Grooving Bit (4389)
5. PCD Radius Bit

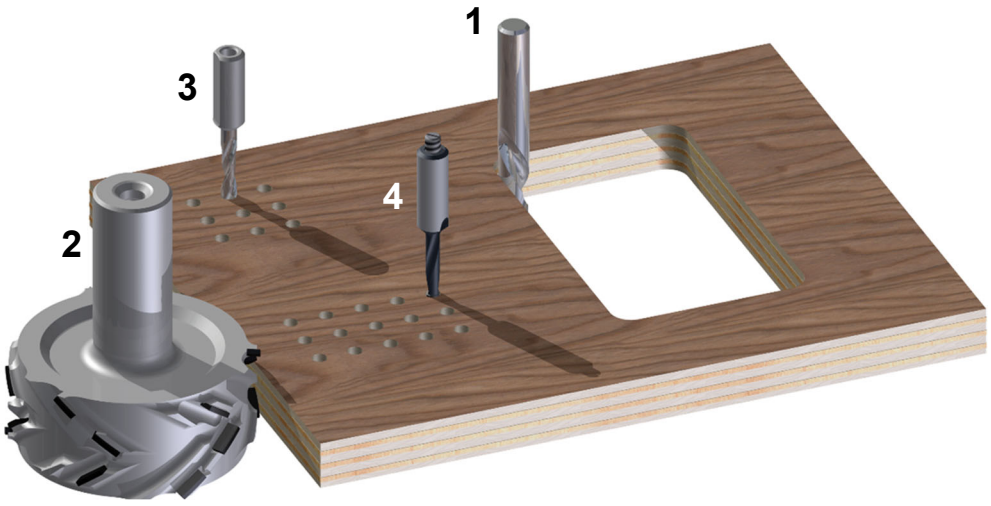
Other Common Tooling Solutions

Machining Aspen Plywood with Paper Laminate

Similar Applications: Birch plywood veneer coated, other paper coated plywood panels

Legend

1. Z3 Compression Bit
2. Gigaspeed PCD Milling Cutter 15700 (or for smaller diameter 6814)
3. Solid Carbide Dowel Drill
4. Carbide-tipped Dowel Drill

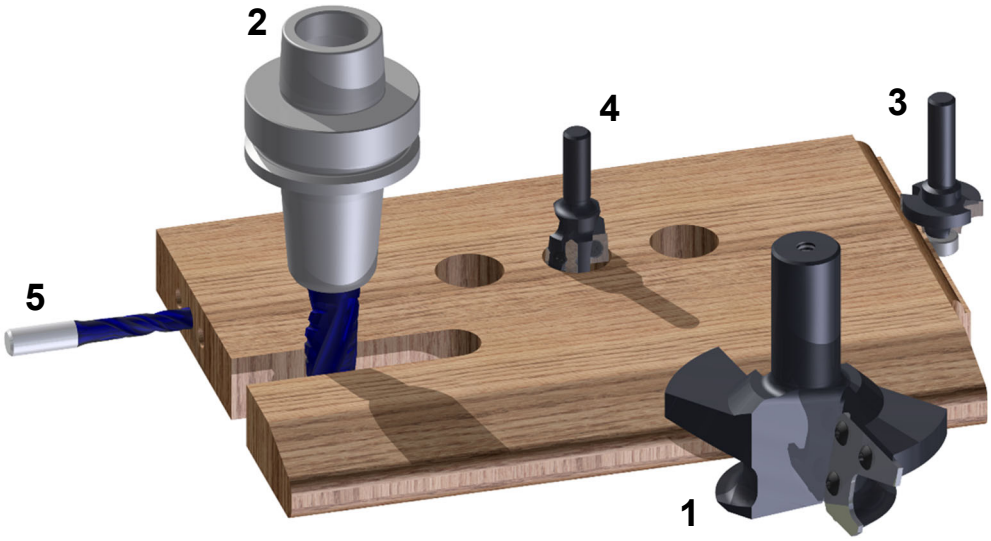


Machining Solid Wood (Oak)

Similar Applications: Various solid wood species, particularly hardwoods and glue up panels

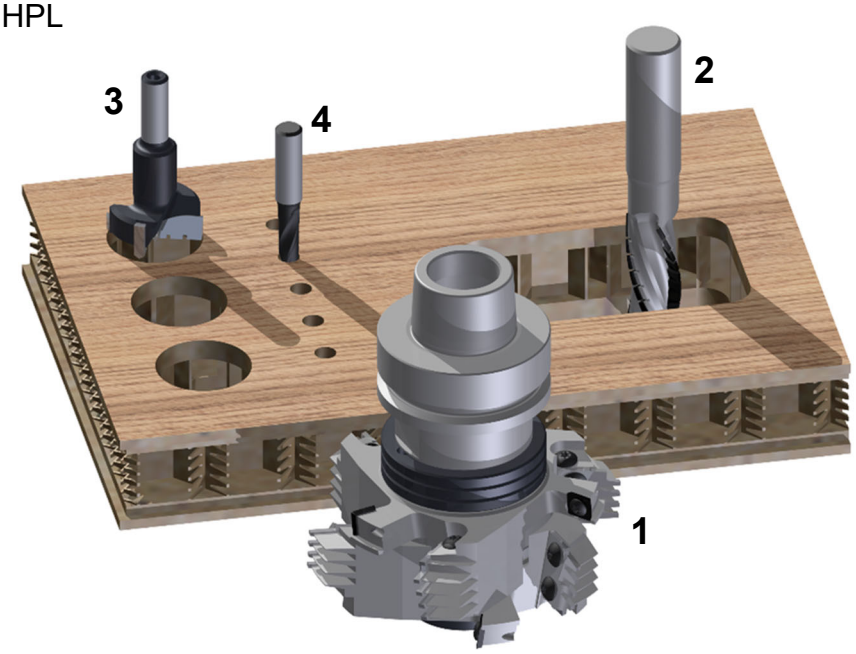
Legend

1. Insert Edge Profile (22530-97)
2. Roughing Cutter Z3 on Heat Shrink Holder
3. Carbide Roundover Bit
4. Carbide Insert Boring Bit
5. Carbide-tipped Dowel Drill Bit



Machining Lightweight Veneered Honeycomb Panels

Similar Applications: Plywood and layered wood panels of different wood species, laminated with HPL

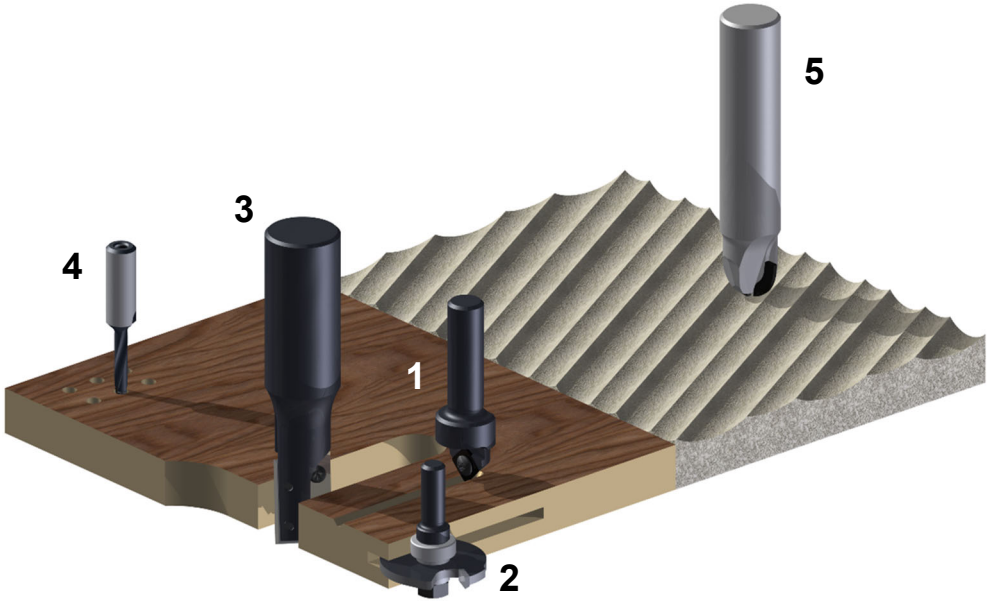


Legend

1. Fourcut 3+3 Cutter Set for Lightweight Panels (custom)
2. Z3 PCD Router Bit (15551)
3. Z2 + V2 Boring Bit with Custom Grind
4. Carbide-tipped Dowel Bit

Machining MDF/Veneer (Left) Cement Fiber (Right)

Similar Applications: MDF and HDF panels, gypsum board panels



Legend

1. PCD Engraving/Grooving Bit (4389)
2. PCD Slotting Cutter
3. Insert Router Bit Z2
4. Solid Carbide Dowel Drill
5. PCD Cove (Ball Nose)

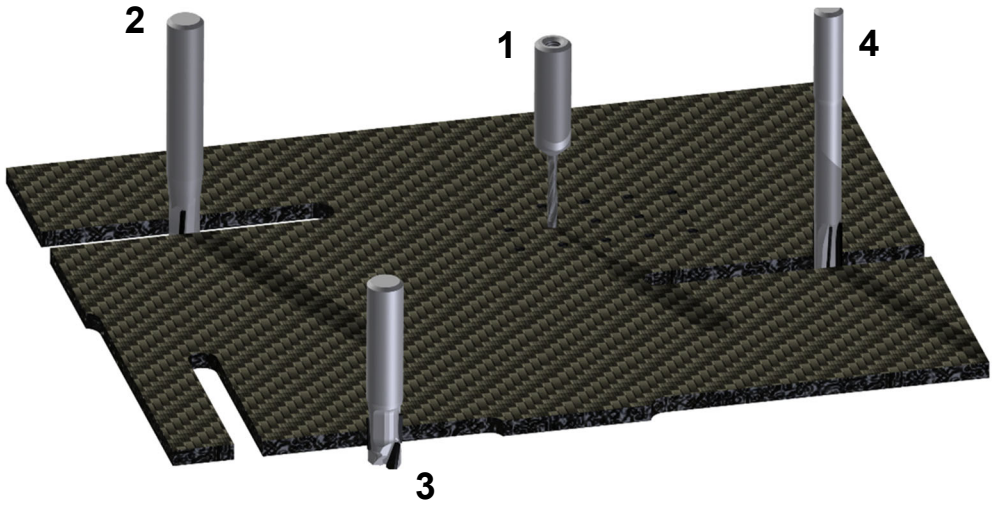
Other Common Tooling Solutions

Machining Carbon Fiber Composite Material

Similar Applications: Fiber-glass reinforced panels

Legend

1. PCD Dowel Bit Z2
2. PCD Router Bit Z2
3. PCD Dual Cut Router Bit (Custom)
4. PCD Z4 Opposite Shear

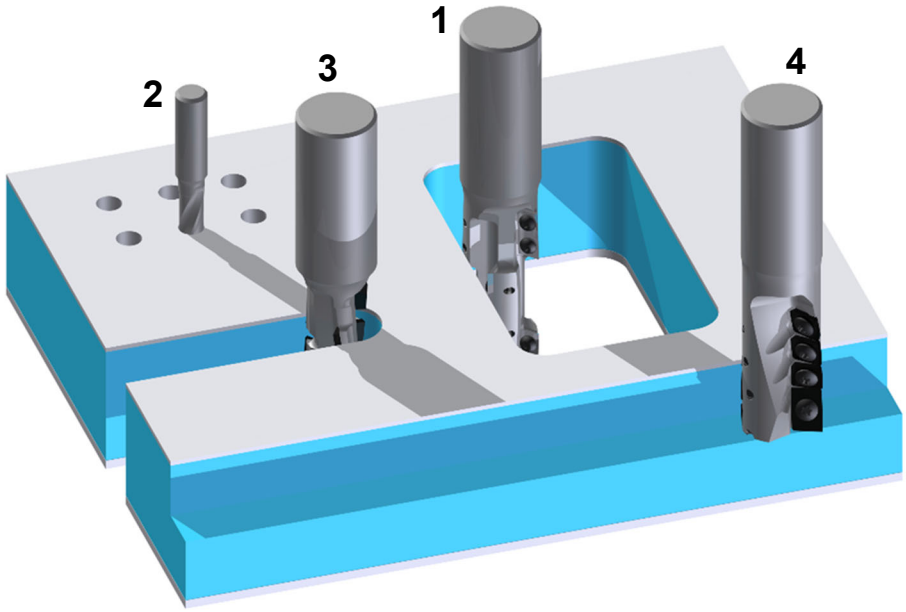


Machining Polyurethane Foam (PU) with Aluminum Laminate

Similar Applications: Polyurethane panels with other laminate coatings top and bottom.

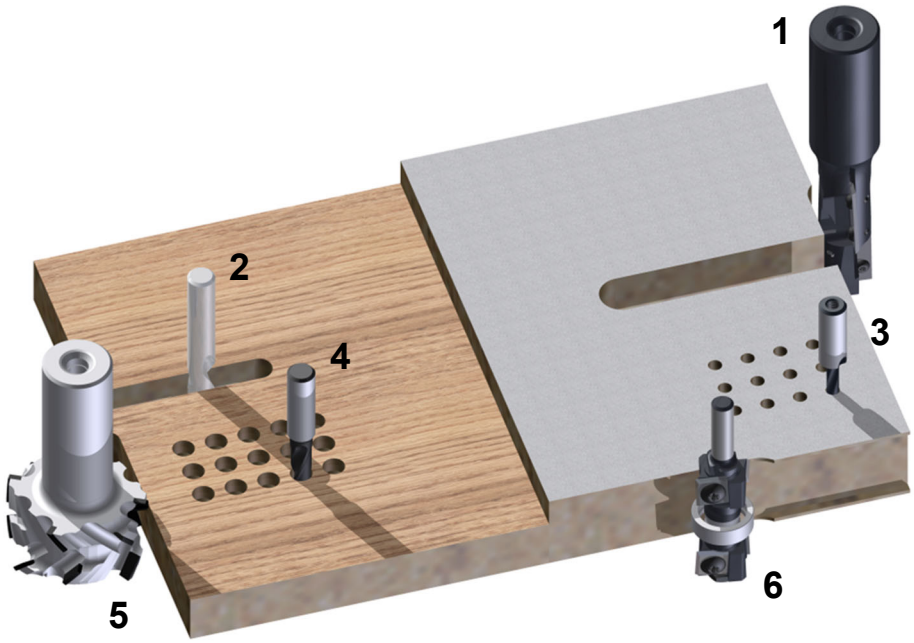
Legend

1. Insert Plunge Router Bit (Custom)
2. PCD Dowel Bit Z1
3. PCD Router Bit Z2
4. PCD Fourcut Roughing Bit



Machining Particleboard with Wood Veneer and Particleboard with Stainless Steel Laminate

Similar Applications: Particleboard panels with other laminates



Legend

1. Insert Router Bit 6243
2. Solid Carbide Compression Bit
3. Carbide-tipped Through Hole Bit
4. Carbide-tipped Dowel Bit
5. Megaspeed PCD Jointing Cutter Z4+2+4
6. Insert Trim Bit (for portable router)

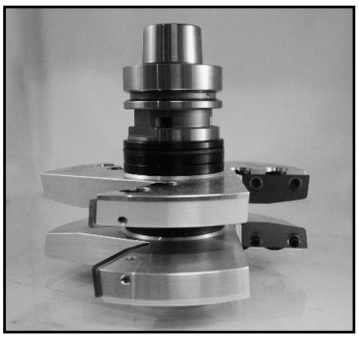
Have specific questions?

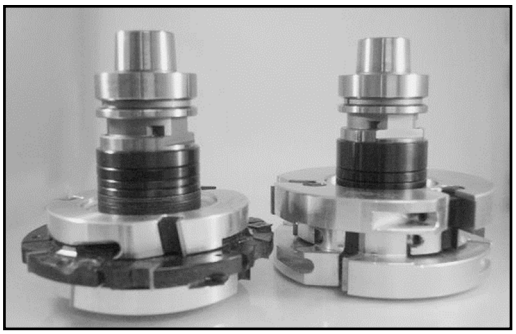
Call us at 1-800-544-8436 or email at info@gdptooling.com

Profile and Custom Tooling Doors, Cabinets, etc.

Profile and Custom Tooling

COPE/STICK TONGUE AND GROOVE RAISED PANEL TOOLING





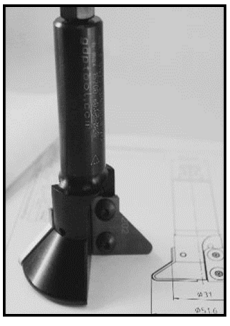
Larger profiles and tool assemblies that can provide flexibility (such as the ability to machine various door thicknesses or tongue width)/ are manufactured on arbor adapters as sets with replaceable insert knives.

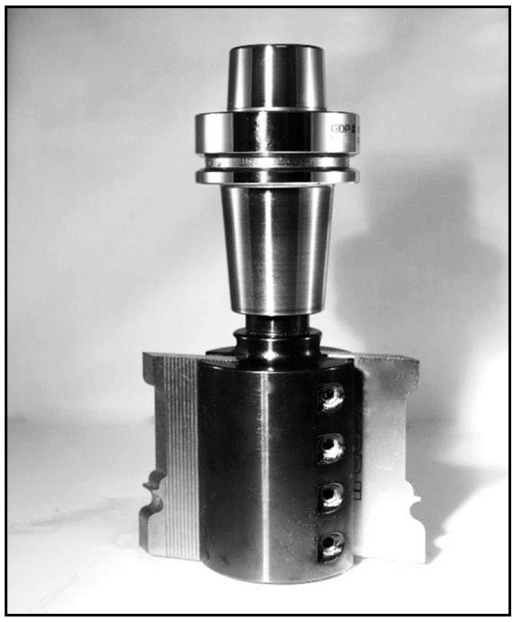
CAUTION: Buyer Beware! Some manufacturers have patented insert tool systems (unique insert blank) that restrict sourcing of replacement knives. This eliminates the ability to price shop!

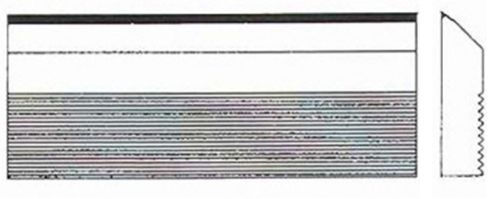
MADE-TO-SPEC TOOLING

Most CNC profile tooling is made-to-order. The process is very simple. A profile drawing or material sample is supplied in order to obtain a firm price quote. Once an order is placed, a dimensioned drawing is submitted for review and revision. Once the drawing is approved, completed to satisfaction, a last approval is needed to proceed.

Larger profile tools are usually made of an aluminum tool body mounted on an HSK arbor adaptor in order to adhere to typical CNC machine spindle weight restrictions.







SHORT RUN PROFILE TOOL FLEXIBILITY

A corrugated knife adapter fitted with a heat shrink tool holder provides flexibility needed for prototype production and one-off projects that won't break the bank. This tool body is made of steel and is available to accept 60 degree corrugated knives of either 8mm, 5/16", or 1/4" thickness.

This tool is designed to accept knives of length 40mm, 60mm, and 80mm corrugated knives.

Similar cutters are seen in the market with an aluminum body, but we strongly suggest you steer clear of such cutters, as the aluminum corrugations in the head will eventually wear out, and the tool can become a real danger to operate!

HELICAL PLANING CUTTER FOR LARGE REBATES, JOINTING, AND TRIMMING IN 5-AXIS MILLWORK OPERATIONS

- Excellent finish with solid carbide, 4-sided insert knives resulting in considerable savings compared to solid carbide tool.
- Excellent chip removal and very quiet running due to helical design.
- Perfect solution for many millwork tasks such as arches and window production.
- The cutter features scoring (spur) insert knives on the bottom to cut clean corners in a rebate cut.

Saw Blade and Groover Adapters for a CNC

Saw Blade Adapters

SAW BLADE ADAPTERS FOR A CNC

Depending on blade diameter needed, there are several options for use of sawblades on a CNC machine. In Figure 1, the shank style adapter is designed for blades up to 8" (200mm) diameter. Figure 2 is a design with saw collars, providing blade stability for up to 16" (400mm) diameter blades. Figure 3 is for blades up to 14" (350mm) and available in several “A” dimension lengths.

Figure 1

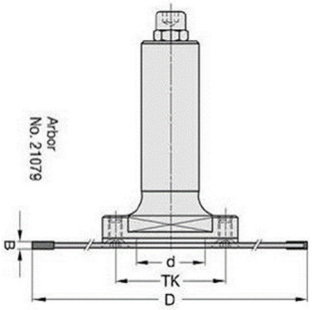


Figure 2

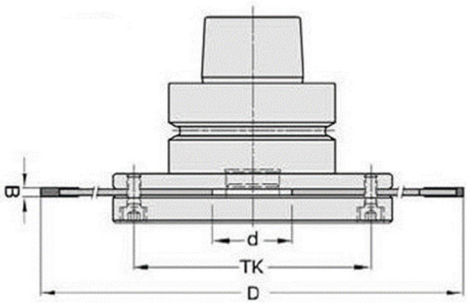
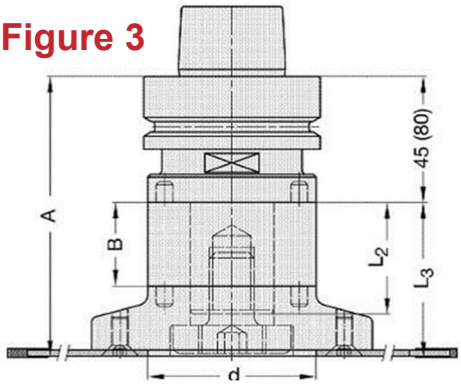


Figure 3



SAW BLADE FLANGE ON HSK-F63 ARBOR ADAPTER FOR BLADES UP TO 14" IN DIAMETER

SAW/GROOVING BLADE ADAPTER FOR BLADES UP TO 8" DIAMETER

AGGREGATE HEADS

An aggregate head can provide 5-axis capability with boring/sawing outputs that facilitate horizontal boring and vertical sawing/grooving. Single, double, and even four outputs are possible on the same head. Torque arm connections are machine specific and must be verified at the time an aggregate head is ordered.

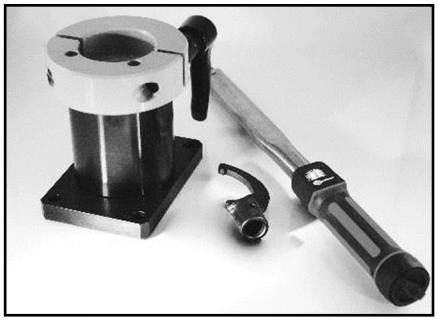
GDP aggregate heads are manufactured in Germany and adhere to the highest standards of quality and accuracy!

GDP TOOLING

Presetters and Setup Fixtures

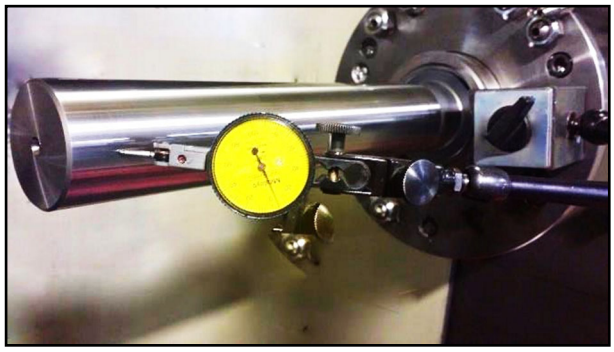
TOOL PRESETTERS AND SETUP FIXTURES

Tool setup fixtures are available in a variety of executions, from the very simple version of a setup tool holder fixture to more expensive and elaborate tool presetters. **Regardless of choice, accurate tool setup will save much time and material waste as well as ensure repetitive accuracy of the final product.**



SPINDLE CALIBRATION BAR

A high precision, **Spindle Calibration Test Bar** (pictured right) allows measurement of spindle accuracy, an important maintenance item of CNC ownership. This is an indispensable tool to test runout and alignment after initial machine installation or relocation, after an unexpected “crash,” or simply as a periodic performance test. This will identify spindle issues long before they become bigger and more costly repair items.



Troubleshooting for Tool Life

SPINDLE DUST PLUGS

Spindle plugs (pictured left) serve to protect the spindle motor from dust intrusion when the machine is either being moved or the machine is performing a saw/grooving operation or horizontal drilling application. Preventing dust from getting into the spindle motor will ensure best maintenance practices and reduction of repair cost.

TROUBLESHOOTING

Tell-tale signs on these photos demonstrate how corrective measures are needed to extend tool life. On **Figure 1,** collet marks are visible on the tool shank. This is an indication that vibration is occurring during the cutting cycle, and it contributes to poor tool life, poor finish quality, and it can also cause the tool to break at its weakest point—right below the shank (as seen on **Figure 2**).

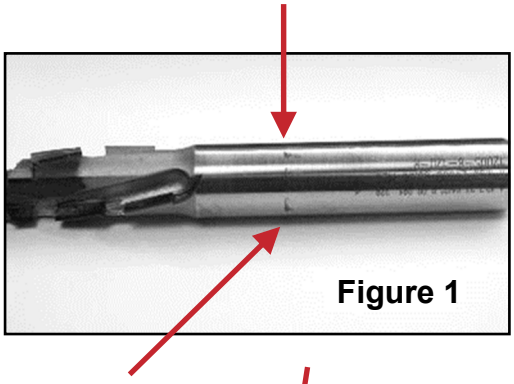


Figure 1

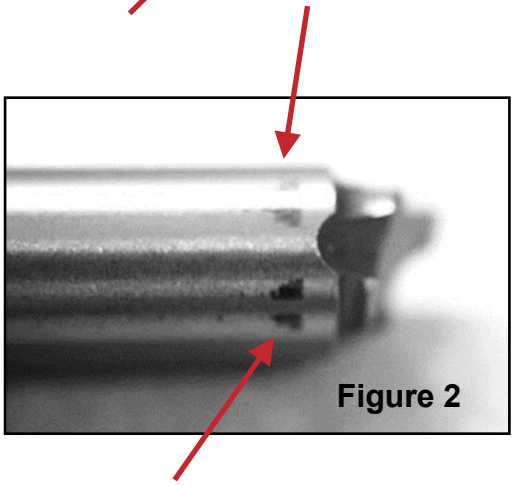


Figure 2

Tool breakage is usually the result of one or a combination of:

- Bad/worn collet
- Poor quality tool holder or collet nut
- Incorrect torque
- Vibration during the cut
- Clamping the tool too high on the shank
- Too shallow a cut in relation to cut length of the tool
- Running the tool when it's dull

Troubleshooting for Tool Life

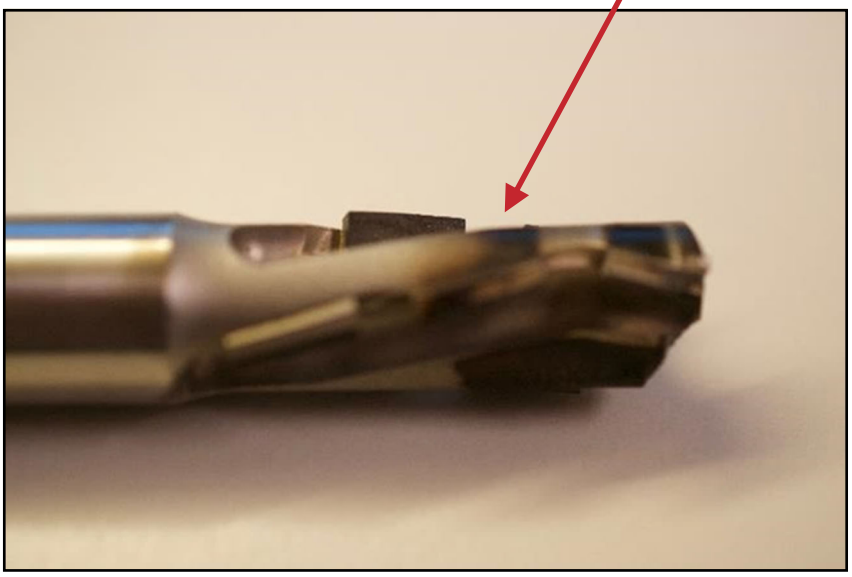
EVIDENCE OF HEAT IN THE CUT

When you see the residue of heat (either black/burned material buildup or a blue discoloration of the carbide or tool body itself), it is time to go over the machining parameters to obtain the correct chip load. This will require one or a combination of:

- Increasing feed speed
- Reducing RPM
- Changing to a tool with less flutes

The picture below shows too much heat generated during the cutting cycle which impacts tool life quite dramatically. When this pattern is seen on a used tool, the chip load is incorrect, and the feed rate most likely needs to be increased—which will increase the chip size.

Evidence of Heat Damage



TOOL BREAKAGE

Possible Cause	Remedy
Deflection—cutting edge too long	Select a tool with cutting edge not more than 1/4" longer than material thickness.
Bad or worn collets	Check tool shank for evidence of chatter marks. When visible, replace collets.
Over-torquing	If tool breaks in the shank area, tightening the collet nut to incorrect torque can cause tool breakage. Invest in a setup fixture and torque wrench.
Overheating	If signs of heat buildup are in the tool flute, check chip load and adjust the feed/speed.
Part movement	Improve vacuum hold-down.
Too much pressure on the tool tip	For rebates, grooves, and slots, use the shortest length to avoid tool breakage.
Tool slippage in the collet	Replace static collet nut with a bearing nut.

We hope this presentation has shed some light on some of the tooling questions and challenges you might encounter and that you can apply some of the information we have shared in your own CNC production!

If we can be of assistance, feel free to call us or email

The GDP Tooling Team
1.800.544.8436
info@gdptooling.com

Notes

GDP TOOLING

www.gdptooling.com

1.800.544.8436 | info@gdptooling.com