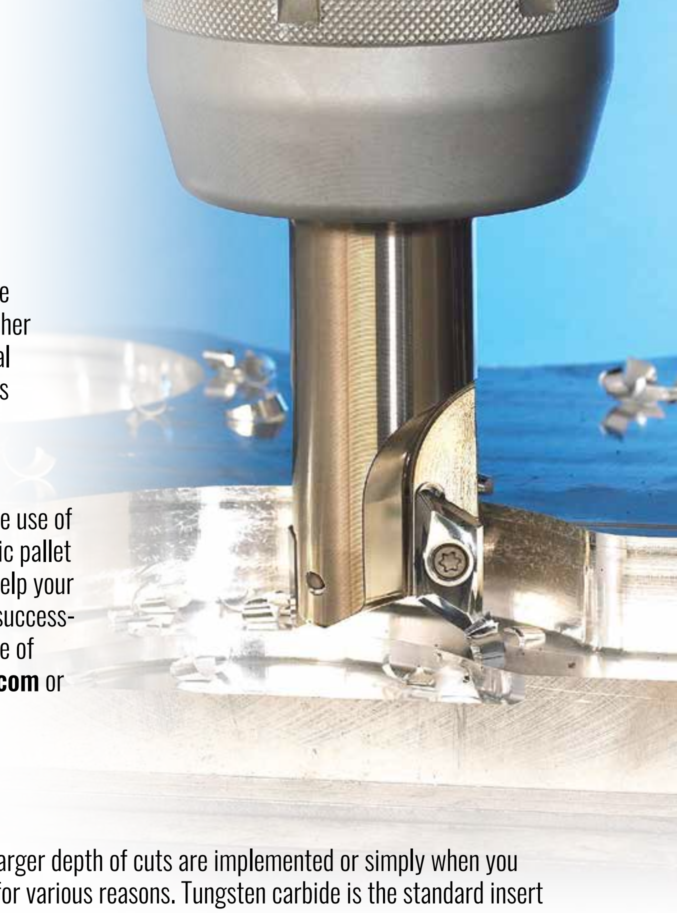


ALUMINUM MACHINING:

Aluminum is abundant, lightweight, low density, corrosion-resistant, machinable, formable, and one of the most widely used non-ferrous metals in the world. Aluminum has different types that are either wrought or cast (e.g. 2011, 2024, 5052, 6061, 6063, 7075, etc.). It has widely become a material common in all shops from prototype facilities, job shops, or high production facilities because of its properties. Aluminum's high machinability rate creates the need to machine extremely efficiently, typically in higher volumes to offset set up costs.

A variety of factors to help maximize productivity when machining aluminum. These factors include use of either a horizontal or vertical machining center, spindle horsepower, CAD/CAM software, automatic pallet indexing, and more. Choosing the right tooling may be the quickest and most feasible solution to help your shop machine aluminum quickly and profitably. Whether turning, milling or drilling, tools that are successful at machining aluminum have common characteristics. Use the information below or contact one of Travers Tool's experienced technical advisors for product selection assistance at **tech@travers.com** or via phone at **800.234.9985**

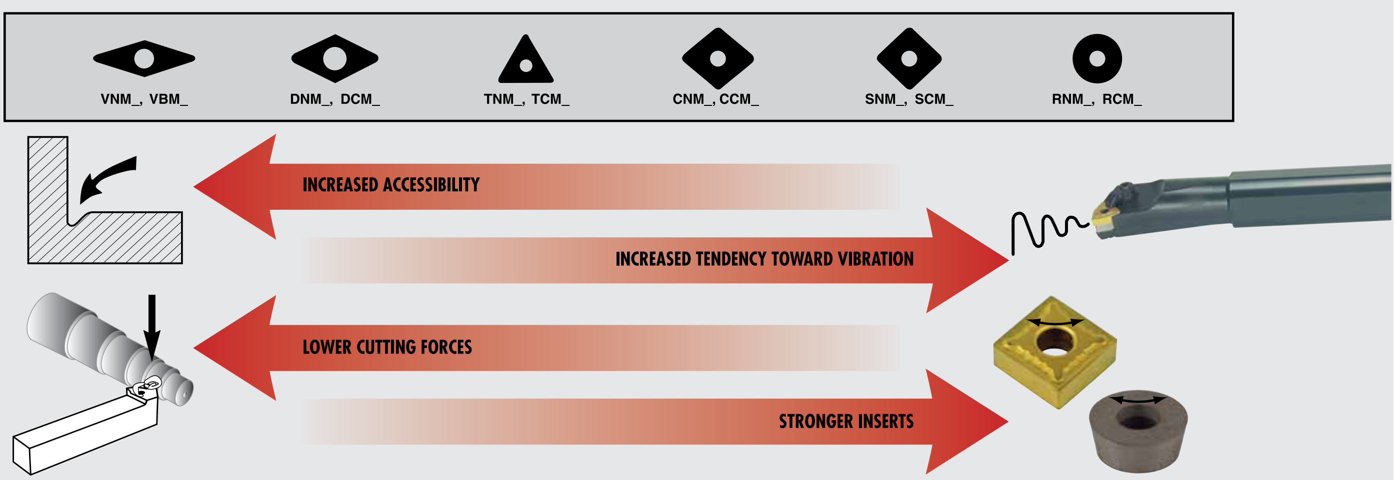
INSERT GEOMETRIES

Indexable cutting tools are normally recommended when larger diameters, larger feed rates, and larger depth of cuts are implemented or simply when you want the convenience of changing inserts vs. regrinding a high speed steel or brazed carbide tool for various reasons. Tungsten carbide is the standard insert substrate and is more brittle vs. high-speed steel. Many inserts have multiple cutting edges (for indexing), have post-processing treatment (such as grinding, polishing or a coating) and may be offered in different radii if the application calls for it. For lathes, indexable tool holder shanks as small as 1/4" can be found and for milling machines, indexable milling diameters as small as 3/8" can be found.

There are several major geometries that have a large effect on an insert's performance: insert shape, rake angles, and the relief angle. Aluminum is a gummy and free-machining material, so sharper angles are normally recommended to help shear through the material versus pushing the material with a honed or dull cutting edge.

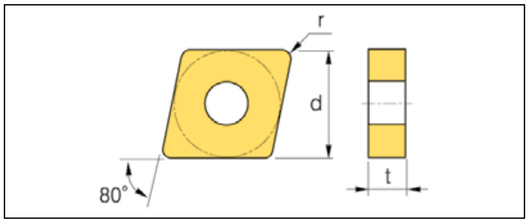
INSERT SHAPE

When producing smaller batches or prototypes, the major determining factor of an insert may simply be what tool holder the shop has available for that application and normally many holders will have specific geometries for machining aluminum. Common shapes such as round, triangle, squares, parallelograms and diamond-shaped inserts can all be used effectively on aluminum. Though when choosing an insert shape, it is normally recommended that the most acute angle be selected after considering the given application, clearance, depth of cut, feed rate and level of interruption that the application will allow. The insert's shape can improve the productivity, effectiveness, and tool life for application. For example, a high-speed finishing application in the single-point turning of aluminum would benefit from using a sharp 35° diamond but inversely you may want to avoid a sharp 35° diamond shape on sand-casted aluminum with a rough surface and use a round-shaped insert.

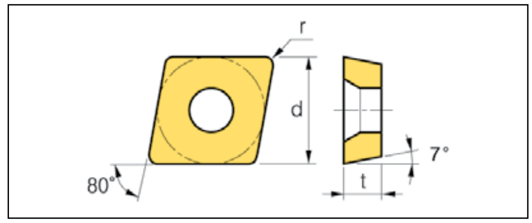


RELIEF ANGLES

The relief angle is the angle between the workpiece and cutting tool, typically on the side of the insert. There are two types of relief angles: positive and negative. Any angle greater than zero is considered a positive relief angle while an angle that equals zero is a negative relief angle. The placement of an insert can alter the overall relief on the holder, for example, an insert with a 20° relief can be tilted to increase its overall relief to 25°. The relief angle works in conjunction with the rake angle to affect the cutting action and its consequence. A positive relief angle is recommended when machining aluminum.



0° RELIEF ANGLE



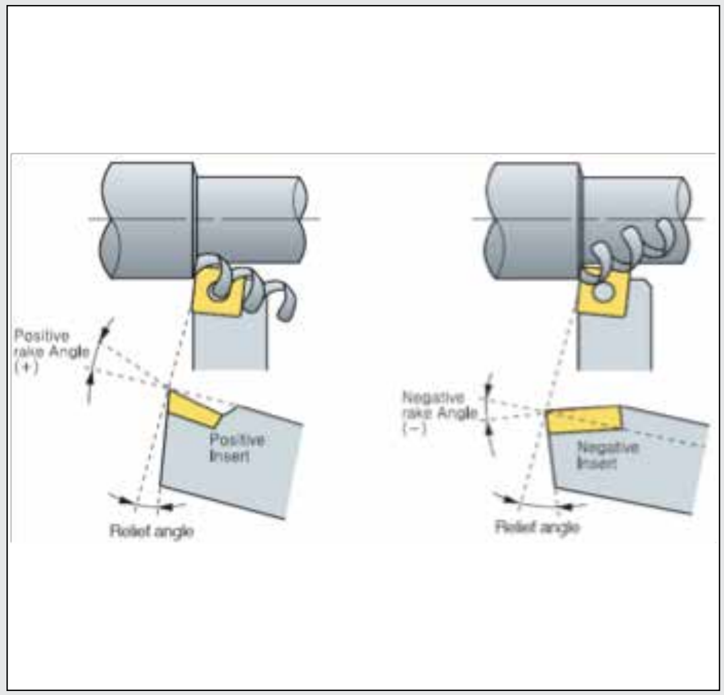
7° RELIEF ANGLE

TWO DIFFERENT RELIEF ANGLES FOR THE SAME SHAPED INSERT

RAKE ANGLES

The rake angle is an angle from the cutting tool tip relative to the workpiece. There are three types of rake angles: negative, neutral, & positive. A positive rake angle is ideal when machining aluminum and most manufactures will use some of their most positive angles for their aluminum cutting geometry. A positive rake angle helps with chip disposal, reduce the insert's cutting resistance, reduce the cutting temperature, decrease built-up edge (BUE), positively affects machinability, and decrease the cutting power required.

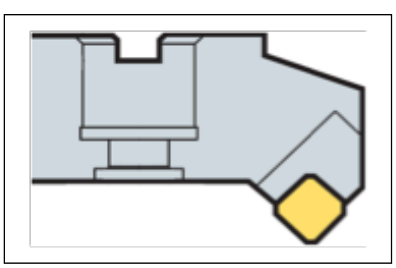
Some caution is advised though, as a positive rake angle may be prone to fracturing and is not recommended on ferrous materials or difficult to cut materials.



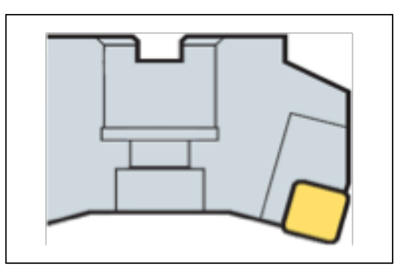
APPROACH ANGLES (MILLING)

Milling tools also offer a variety of additional approach angles. The approach angle is the angle of the insert when it enters a cut (workpiece). The most common of these approach angles are 45°, 75°, and 90°. Each angle provides their own pros and cons but due to the high machinability rate of aluminum, 90° cutters are the most common for non-ferrous specific cutters. A 90° approach angle is most commonly used because of the various applications that can be achieved. Most 45° approach angles are limited to face milling operations and chamfering, while many 90° approach angle milling cutters being able to face mill, shoulder mill, slot mill, ramp, circular interpolate

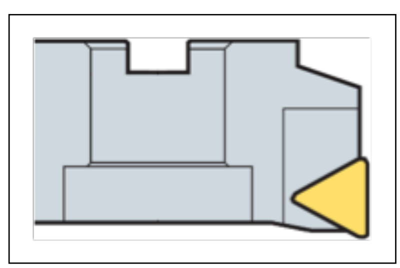
It should be noted that a 90° approach angle does push the cutter radially resulting in possible deflection, increased vibration and chattering while a 45° approach angle will produce a better finish and have higher table feed rates due to chip thinning.



45° APPROACH ANGLE



75° APPROACH ANGLE



90° APPROACH ANGLE

CHIPBREAKER / CHIP FORMER

Aluminum chips are gummy and can quickly become unmanageable. A high positive and sharp chipbreaker is recommended for aluminum. Typically, the rake angles and sharp cutting edges are accentuated with aluminum / non-ferrous specific chip breakers and too sharp for many other materials. Chipbreakers for aluminum are normally wider and engineered to evacuate chips at varying depth of cuts. Proper chip control is essential for high-speed machining, especially in hole making and boring applications. For milling operations, it is very important to evacuate chips in the flutes and near the cutting edge as an accumulation of chips can cause premature tool life. The up sharp cutting edge that evacuates chips is one of the most important features for aluminum cutting inserts.

AK SPECIAL CHIPBREAKER FOR ALUMINUM:

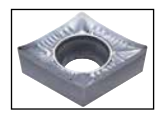
1. **High rake angle & tabby pattern chip pocket:** low cutting load
2. **Unique rake angle design:** effective chip breaking and good chip flow
3. **Unique and 3-dimensional top face:** longer tool life & excellent surface roughness
4. **Tabby pattern & sharp cutting edge:** distributed cutting load, longer tool life
5. **Buffed on top face:** excellent machining & chip flow, reduced built-up edge

SUBSTRATES, COATINGS, & POST TREATMENT

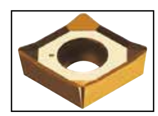
The performance of a cutting tool can be drastically altered by the substrate, surface treatment or coating applied to an insert when machining aluminum. Because of its affinity to aluminum simply changing the substrate of the tool from carbide to a synthetic diamond (i.e. P.C.D.), can exponentially increase the tool life and cutting performance. Though typically when machining aluminum with indexable tooling, a tungsten carbide substrate is recommended. A wear-resistant grade such as an industry-standard C2 or C3 is commonly used. These inserts can be left uncoated and usually have a sharp cutting edge with a minimal hone and for milling inserts an "F" style edge prep (e.g. SECN42AFFN) is applied. It is recommended that you try to utilize a ground tolerance vs. a molded tolerance (e.g. CCGT vs. CCMT or APHT vs. APKT). A wear-resistant substrate, sharp cutting edge, ground tolerance, and a post-treatment such as a polished surface can result in an insert that can perform better: run faster, produce a better finish, extend tool life, and resist built-up edge.

To add additional performance and longevity, an insert can be polished, or a coating can be added. Many high-quality inserts that are designed for aluminum machining are polished instead of being coated. Though coatings can also be applied to inserts to add hardness, lubricity and extend tool life. It is recommended that the coating be compatible with aluminum and generally a thinner type. Typically, thinner PVD applied coatings are recommended. The major benefit of using an uncoated polished insert is a cost-effective solution for many of the current holders that you may have in your shop today. Here are some of the different coatings and substrates that are offered in the market today:

- **Uncoated Polished Surface:** A post-process that buffs the surface of an uncoated surface to reduce friction. Not a coating but the most common type of post-process for non-ferrous specific carbide inserts. The mirror-like finish improves surface finishes and tool life vs. a standard uncoated insert. These inserts are economical for shops, they are cost-effective and easy to identify. In a job shop settings or non-production type of settings, these should be your first choice.

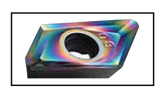


- **TiN (Titanium Nitride Coating):** popular PVD applied gold coating found on many cutting tools. This coating increases lubricity, wear resistance, thin, improve tool life in dry conditions and is very cost-effective. A slight improvement over a polished surface (when applied to a polished surface) but used primarily because it's cost-effectiveness.

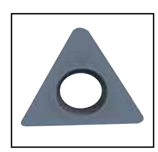


- **TiB2 (Titanium Diboride)**: a PVD applied ceramic-based coating that is becoming more popular with carbide cutting tools. This coating does not chemically react with aluminum for added lubricity when machining aluminum. Along with the increased lubricity, the coating improves tool life and hardness. This coating is more effective than a standard TiN coating but may not be available in many inserts.

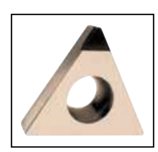
- **DLC (Diamond Like Carbon):** a PVD applied nanocomposite amorphous carbon coating. This coating has very high hardness, is extremely resistant to wear, low friction rate, extremely slick, and performs better during dry operations. This coating improves surface quality, burr formations and the thin coating helps maintain a sharp cutting edge. It performs better than a TiB2 coating & is more cost-effective than a diamond coating.



- **Diamond Coating:** A CVD applied a coating of a pure crystalline diamond. This coating is thicker than a DLC coating, dramatically improves hardness, extends tool life, and enhances abrasion resistance. This coating outperforms all other coatings in terms of applicable speed and tool life. One of the main benefits of a diamond coating is that it can be used existing tool geometries, meaning multiple cutting edges and the use of current holders. Diamond coatings, however, are not solid diamonds and can have some inconsistencies to them, like other coatings and it is not recommended for re-sharpening. These coatings can be used on green carbide and ceramics as well.



- **PCD (Polycrystalline Diamonds):** super-hard synthetic diamonds that are brazed onto carbide tooling. These tools have the best performance of all cutting tools (less solid natural diamonds) on non-ferrous materials. PCD tooling has an extremely low affinity with aluminum and can last days if not weeks in a full production environment. The tool life for these tools is very predictable. These tools have a sharper cutting tool edge than a diamond-coated insert, producing a better finish. The major drawback to these tools is the limited geometries and the price point. Justification for these tools can be difficult unless in a production environment, as these tools can be expensive.



There are other coatings that are applied on inserts, some that are not as commonly seen that work well on aluminum such as ZrN & TiCN and some coatings are not as effective. When possible, try to avoid coatings that contain aluminum on their top layer such as a multi-layered CVD applied Al2O3 (aluminum oxide) or PVD applied TiAlN or AlTiN coatings. A top layer of coating that contains aluminum may accelerate built-up edge (BUE) and cause premature or catastrophic failure. There are exceptions to this rule, such as a TiAlN coating that is applied with the HiPIMS (High Power Impulse Magnetron Sputtering) method.

COMMON ISSUES AND INSERT FAILURE

One of the most common problems when machining aluminum is built-up edge (BUE). Built-up edge is when the metal that is being cut will accumulate on the rake face of the insert. This accumulation dulls the cutting edge, degrading the tool life and leaves a poor surface finish. There are several means of preventing BUE, which include using sharp geometries, using an aluminum wear-resistant specific grade, utilizing a polished surface or coating, utilizing ample coolant and applying the correct speeds. Though coolant tooling is recommended if not available ample flood coolant.

Chip evacuation is another common problem when machining aluminum. This issue is more commonly seen with solid round tools but can be seen when machining aluminum in high volumes, at high speeds. Milling tools with too many teeth can 'pack' chips and this can lead to tool failure. Wide insert pockets with plenty of clearance for chips to evacuate is recommended. Utilizing too many teeth is not recommended, while coarser teeth selection with higher feed rates.

CHOOSING AN ALUMINUM CUTTING MILLING TOOL

When choosing a milling tool, there are a wide variety of milling strategies and choices in tooling to consider when approaching an aluminum application. Like most situations in a shop, the right choice depends on the application. Asking yourself the right questions at the beginning of the process will help your shop mill aluminum more efficiently and succeed in achieving high productivity and profitability.

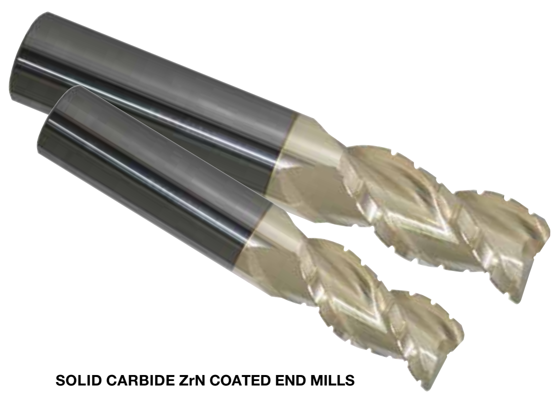
KNOW YOUR MACHINE TOOL...

The machine tool is one of the largest factors and a great place to start with choosing the most efficient milling tool for your aluminum application. Different machine tools will dictate what milling strategy and what type of tool to utilize. Situations such as machines with power limitations, utilizing quick rapid motions in a quality CNC machine, or plowing through aluminum with a sturdy machine that has box ways with a dual contact CAT50 spindle would all dictate how best to utilize your machine spindle.

HIGH SPEED STEEL END MILLS

MANUAL MACHINE

Manual Machines are great for quick, simple set-ups. A good amount of maintenance work, one offs and repair jobs done on aluminum are on manual set-ups. Though it's important to note, utilizing some of the newer tools with the slower speeds and feeds can be overkill on your set-up cost. High speed steel tooling is a great option for one offs and machines with lower speeds and manual or traditional power feeds. The toughness in the tooling is more forgiving to sudden feed rate changes and has a very attractive price point. For larger quantities, longer tool life, and for higher speeds, solid carbide is a great choice (it is recommended that carbide is used with constant feed). Although higher in price, solid carbide end mills are more productive and offer a larger selection in geometries and coatings. Generally, for anything larger than 5/8", an indexable milling cutter should be considered.



SOLID CARBIDE ZrN COATED END MILLS

CNC MACHINING CENTER

A CNC machining center is a very cost-effective way to machine aluminum. Because of the large variety of machine tools available, the selection of tools can become overwhelming. Some factors that you may want to consider when choosing a tool are the power consumption, tool paths generated by your CAM (or conversational), and machine feed rates. CNC machine tools are great for roughing complex geometries quickly. When roughing, we ideally want to remove the most amount of material as quickly as possible and modern-day machine tools do not necessarily need a large cutter diameter to achieve high metal remove rates. Utilizing high machine speeds, machine feed rates and a constant chip thickness through H.E.M - high efficiency milling techniques (large axial, low radial depth of cut, and high feed per tooth) and adaptive milling paths (a.k.a. dynamic or tricordial milling paths) allow the use of smaller diameter tooling that remove material faster than traditional milling methods (larger radial depth of cuts and smaller axial depth of cuts). Solid carbide end mills that are specially designed for non-ferrous materials excel in these applications. Special geometries help evacuate chips and aggressive chipbreakers lower tool pressure and allow for higher feed per tooth. Coatings such as TiB2, ZrN and Diamond coatings provide excellent tool life and boost speed capabilities over uncoated solid carbide end mills and high-speed steel substrates. As for applications that require or can fit a larger diameter tool, indexable milling tools provide a great solution. Most major manufactures understand the need for high speed machining in today's market and develop tools with many features and benefits to meet these demands. Milling cutters that are specifically designed for high speed machining in aluminum with long cutting edges, sharp rake angles, polished surfaces or advanced coatings, PCD brazed tips, and other geometries that support high speed machining are very common in the market.

INDEXABLE SHELL/FACE MILL

SOLID VS. INDEXABLE TOOLING

Solid carbide and indexable tools have their pros and cons. Choosing the correct tool for the job can be easy at times, such as using an end mill to finish the floor of a 1/2" pocket or using an indexable face mill to deck off the top of a part. Then there are applications where choosing a tool can become more complicated because of the different CAM software + machine tool capabilities. A large determining factor when deciding what tool to use for a roughing strategy is how much metal you can remove or your "material removal rate". Material removal can be quantified by using the following formula:

Material Removal Rate (M.R.R.) = (Radial Depth Of Cut) x (Axial Depth of Cut) x (Feed Rate)

So if we machined a piece of aluminum, assuming all factors except the tool parameters are the same, with a five flute 1/2" solid carbide end mill and a two flute 1.0" indexable end mill, we can see that the metal removed with the indexable tool is over twice that achieved by the solid carbide tool:

Diameter	IPT	RPM	# of Teeth	Inches per Minute	DOC	Radial Engagement	Cubic inches per min
0.5"	0.005"	7600	5	190.0	1.0"	0.025"	4.750
1.0"	0.007"	3800	2	53.2	0.6"	0.350"	11.172

So, for roughing applications, we want to use the largest possible cutter that the application or machine tool will allow without chattering or stalling. This is one of the reasons why the machine tool is so critical! Large taper spindles with secure mounting such as dual contact are great for taking larger radial and axial depth of cuts without creating chatter. Small tools with a large radial and axial depth of cut may create chatter, limiting the engagement of the tool. Part designs also may be complex and CAM programs would only engage a 1.0" tool for a minimal amount of time.

WHERE SHOULD YOU CONSIDER USING AN INDEXABLE MILLING CUTTER VS. A SOLID CARBIDE TOOL?

This has many variables and while each situation should be assessed on a case by case basis, there are two factors that come to mind when deciding when to use an indexable tool over a solid carbide tool – price and reliability. The replacement cost of a solid carbide tool over the 1/2"-5/8" range can start getting expensive. Indexable tooling should start being considered at this point. While indexable tooling starts at diameters as little as 3/8", you may not always see the cost benefits on diameters this small. Cutting diameter sizes starting somewhere in the range of 5/8"-3/4" is where you start seeing good financial benefit.

The initial dollar value, however, may not be your determining factor. Process reliability such as chip management, tool changes, or taking advantage of a more reliable (sturdy) set-up may outweigh the benefits of aforementioned cost savings. Solid carbide end mills can create small, difficult to manage chips that can get into areas such as way covers, causing damage or requiring additional maintenance time. However, chips created with indexable tooling (due to the feed rate and depth of cuts) tend to be heavier and more manageable. Tool changes can be a determining factor as well. Although aluminum tooling with advanced coatings tend to have exceptional tool life, the ability to change insert corners after a certain lot or quantity for process reliability may also be a determining factor. Indexable tooling also offers multiple radii, coatings or substrates (such as PCD) to potentially add value to your application. Refer to the pros and cons on solid carbide and indexable tooling below, to help you weigh your decision.

Solid Carbide End Mills	
PROS	**CONS**
Precise, most high quality endmills are ground within several 0.0001"	Brittle, if you crash or chip you may have to replace the whole endmill
Can produce amazing finishes	Limited feed rate capability
Small Diameters	Limited size range
Low tool pressure	Setting up tools with minimal runout can be time consuming or costly.
Can be reground	Limitation to regrinding
Relatively low replacement cost on smaller diameters.	Large diameters are very expensive and leave little room for mistakes
Can use used endmill shanks as a reference pin	Can create small unmanageable chips
Increased speed due to rigidity	

Indexable Tooling	
PROS	**CONS**
Most inserts come ground but holders.	Holders tolerances are generally looser
Can incorporate PCD inserts for mirror finishes	PCD inserts are expensive
Inserts can be indexed and get multiple cutting edges from one insert	Initial set up costs are expensive
Insert costs are relatively lower.	Most companies sell inserts in multi piece packs.
Can take more load and interruption	Higher tool pressure
Capable of higher feed rates and depth of cuts	Requires more power to run
Manufactures can make a large diameter cutters	Limitation on smaller cutters
Thicker more manageable chips.	

SOLID CARBIDE ZrN COATED END MILLS

INDEXABLE FACE MILL

Contact one of Travers Tool's experienced technical advisors for product selection assistance at **tech@travers.com** or via phone at **800.234.9985**