FEBRUARY 2005 / VOLUME 57 / NUMBER 2

➤ BY KEN KING, THINBIT/KAISER TOOL CO. INC.

ADDED FUNCTIONALITY

Groove-and-turn tools perform multiple operations while occupying just one tool position.

All images: Thinbit

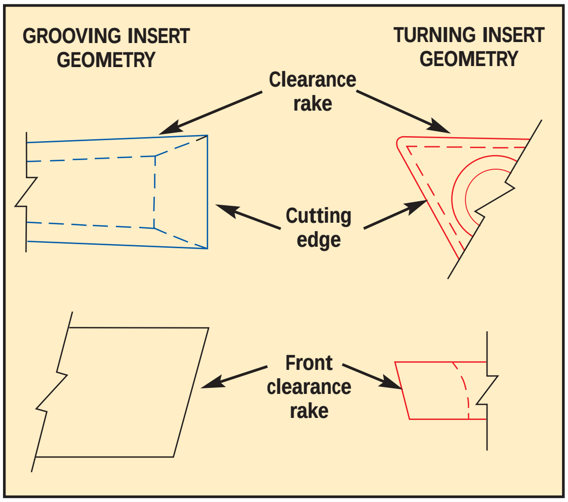
Groove-and-turn tools help overcome lathes' limitations.

The versatility of many lathes is offset by the limited number of tooling positions they offer. Multifunction tools can remedy this shortcoming. They allow two or more machining operations to be consolidated into a single tool position. Multifunction tools also decrease cycle times, lower tool inventories and increase programming flexibility.

A common multifunction tool is one that grooves and turns. Grooving-and-turning inserts are capable of grooving, turning, and profiling arcs and angles. There are few limits on the part geometries that can be machined with this type of insert. And, it provides benefits for jobs ranging from single-part prototyping to long production runs.

How They Work

Groove-and-turn tools feature geometries that allow them to cut axially and radially, as well as toolholders and clamping systems rigid enough to handle the cutting forces from both directions.



A groove-and-turn insert is a hybrid of a plunge-grooving insert and a turning insert. Standard grooving inserts have a straight cutting edge—for machining the bottom surface of the groove—that tapers inward behind the cutting edge and a clearance rake angle below the cutting edge. A standard turning insert has an angle of less than 90° behind the cutting corner and a clearance rake angle below the cutting edge.

With a groove-and-turn tool, the straight edge for grooving and the tapered edge for turning are the same. In order for the insert to perform dual functions, it must be run at a sufficient DOC to deflect the cutting edge slightly (Figure 1). This action creates a clearance angle behind the cutting corner. It also helps to form and break chips.

Another geometric consideration is the clearance rake angle below the cutting edge on a turning insert. This clearance angle is not required on a plunge-grooving insert. The addition of it allows a grooving insert to turn, but at the cost of decreasing the cross-sectional area of the insert.

The rake angle on a true turning insert is typically in the 7° to 14° range. An angle this large would unnecessarily weaken a multifunction tool, so an angle of 3° to 5° is generally found.

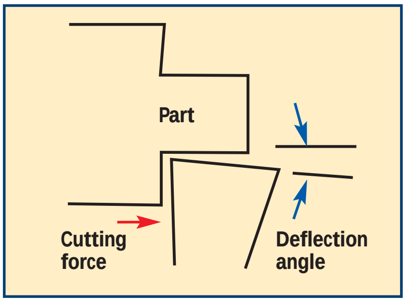


Figure 1: A groove-and-turn insert must be run at a sufficient DOC to deflect the cutting edge slightly, in order to create a clearance angle behind the cutting corner.

Multiple Choice

Listed below are five steps for selecting a groove-and-turn tool. (These steps also can be followed when choosing a tool for OD/ID grooving and face grooving.)

1. Review all of the grooves to be machined for a specific job. Note the narrowest groove width and deepest DOC. If the deepest part DOC is shallower than the DOC of the narrowest grooving insert, a single groove-and-turn insert can cut all the grooves. (A grooving insert can cut grooves larger than itself.) If the deepest part DOC specified exceeds the depth of the narrowest groove, either consult a manufacturer about having a special insert made or select two or more inserts that cover the range of grooves to be produced.

If there are dissimilar part diameters or shoulders at a groove location, the deeper DOC can be ignored by using a shoulder-style insert. Selection of this type of insert is based on the shallower DOC and insert/toolholder orientation. A shoulder-style insert can cut grooves regardless of whether or not the part has shoulders (Figure 2). Special consideration is required for grooves that have different radii, as is explained in the next step.

2. Different-sized part radii can be formed with an insert able to produce the smallest radius required. An insert can form a part radius larger than the radius on its cutting edge. Only equal, or larger, part geometries can be cut or formed, not smaller ones. Review all the radii on the part, in the grooves and at any transitions in diameter. Be aware that the form left by the insert will match the geometry ground on it and that all of the changes in diameter and all the grooves will have, at minimum, the radius of the insert.

A radius-style insert delivers a better finish and lasts longer than a nonradius tool. The reason is because all cutting edges wear, but sharp edges wear faster than radiused edges. Finish improves as radius size increases and the feed rate remains the same.

There are cases when grooves of about the same width cannot be cut with the same insert. An example would be a 0.062"-wide groove with sharp corners and a 0.062" groove with a full radius. If a sharp-corner insert were applied to a full-radius groove, it would overcut the bottom surface. Conversely, a full-radius insert would not form the requisite sharp corners.

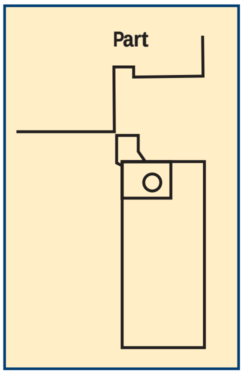


Figure 2: With a shoulder-style insert, the outer corner of the cutting edge lies outside the line of action of the toolholder. This allows the part diameter on the outer corner of the cutting edge to be much larger (right-hand side of illustration). Practically speaking, the only limit on the OD is the distance the toolholder extends from the machine.

3. Facing operations and changes in part diameter require special consideration and, possibly, a consultation session with a cutting tool applications engineer or a CAD programmer. Either, with the aid of a CAD system, could determine all the points where the insert or toolholder might crash into the part. They also could identify those part features that would prove most problematic to machine and recommend the best insert for the job. Sometimes, though, it is not possible for a single insert to produce every part feature.

The position of the toolholder system must be considered, too. Sufficient clearance is required for the toolholder components, relative to the part geometry. A shoulder-style insert can resolve certain clearance issues.

4. Many options are available for forming and controlling chips. The most important thing to remember is that the insert cuts axially and radially. Therefore, the chip-control geometry selected must address chip flow in both directions. It is also important to note that even a neutral geometry will successfully control chip flow in most materials, if the proper speeds and feeds are selected.

5. All manufacturers offer speed-and-feed and coating recommendations for common workpiece materials. However, the complexity of applying multifunctional tools can overwhelm the simple charts and graphs printed in many manufacturers' catalogs. The assistance of an applications engineer may be required.

Path Programming

After the tooling is selected, the next step is to program the cutting paths. Certain characteristics of groove-and-turn tools must be taken into account when programming. For example, a groove-and-turn insert may not be able to withstand the cutting forces that a true grooving or turning insert could. Also, consideration must be given to the order in which operations—grooving, turning, chamfering, and the profiling of radii and angles—are performed. It's recommended that part geometries be produced in the following order:

1. *Rough-turn diameters, arcs and angles.* Multidirectional turning is an efficient process for this phase of cutting. It involves plunging radially to the recommended cutting depth, turning axially toward the chuck, plunging radially to a recommended cutting depth

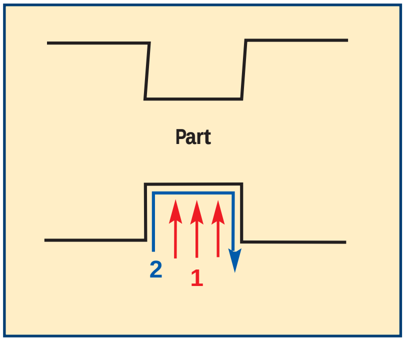


Figure 3: For grooves less than three times the width of the insert, plunge into the material at the center of the groove and stop just before the groove depth is reached (1). Then cut down the left and right walls to the same depth and follow with a sweeping pass around the perimeter (2).

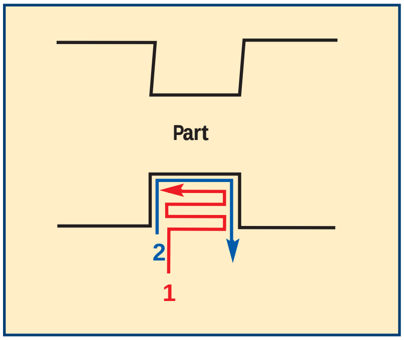


Figure 4: For grooves three or more times wider than the insert, rough-turn to remove the majority of the material (1) and then make a finishing pass (2).

and then turning axially away from the chuck. This method keeps the insert in constant contact with the part, which maximizes the material-removal rate.

Not all CAD/CAM systems support multidirectional turning and, therefore, manual programming sometimes will be necessary. Rough turning leaves steps at the transition points that are removed during the next operation.

2. *Plunge-groove.* Depending on the width of the groove relative to the insert, one of two methods can be utilized. If the groove is less than three times wider than the insert, plunge into the material at the center of the groove and stop just before the groove depth is reached (Figure 3). Next, cut down the left and right walls to the same depth. Finally, make a sweeping pass around the groove perimeter. During this pass, add any chamfers and finish the radii within the groove. This pass also pushes burrs outside the groove.

The second method is for cutting a groove that is three or more times wider than the insert. Rough-turn to remove the majority of the material and then make a sweeping finishing pass (Figure 4).

Standard grooving practices, such as peck cycles and dwells, should be applied with both methods, as dictated by the workpiece material.

3. *Finish-turn.* A lighter DOC can be used to finish any geometry roughed out during the previous steps. This is also a good time to remove or reduce the size of burrs.

Some Drawbacks

While the benefits of groove-and-turn tools are many, there are drawbacks. For example, it is difficult for a manual machine to take full advantage of the profiling capability of a groove-and-turn insert because profiling requires precise control of two axes simultaneously.

Also, tool life may be affected by uneven wear on the cutting edges because of the variety of operations and the amount of material removed with a single insert.

All things considered, though, the benefits of groove-and-turn tools outweigh their drawbacks. As with any newer technology, there is a learning curve. But once users understand how to apply these tools, the overall efficiency of their machining operations increases dramatically. △

About the Author

Ken King is operations manager at Thinbit/Kaiser Tool Co. Inc., Fort Wayne, Ind. For more information about the company's products, visit www.thinbit.com, call (888) THINBIT