

# Top Tools

Top five cutting tool solutions to boost a moldmaker's competitiveness.

## cover story

By Alan Richter, Editor

Hypercompetitive accurately describes the moldmaking industry. To remain competitive, moldmakers must reduce lead times by implementing the best manufacturing processes—quality is a given—and that requires having the proper equipment. “In today’s economy, unless you reinvest in your company, you aren’t going to make it,” said Doug Kline, national sales and general manager for the consumables and tooling group of distributor Single Source Technologies Inc., Auburn Hills, Mich.

The total machining system, or package, is important, including the machine tool, fixturing and accessories, but like a racecar where the tires determine the difference between victory and defeat, cutting tools are critical. “It all happens at the tip,” Kline said.

Presented here are five top cutting tool solutions for moldmakers: deep-hole drills that avoid pecking, high-feed cutters, tools for machining ribs to minimize or eliminate EDMing, high-accuracy cutters and PCBN cutting tools. *[Editor’s note: CTE refers to the cubic boron nitride used for cutting tools as polycrystalline cubic boron nitride (PCBN) and the superabrasive material for grinding wheels as CBN. Others often refer to PCBN as CBN when used for cutting tools and those references were left intact.]*

## Learn more about moldmaking



An additional multimedia presentation in connection with this article is featured as an Interactive Report on [www.ctemag.com](http://www.ctemag.com). Also, read more commentary on moldmaking by visiting Alan Richter’s Web log in the CTE Community section.

## Top Tools *(continued)*

### Going Deep

Plastic-injection molds typically have holes for water lines and ejector pins. Traditionally, when drilling more than 4 diameters deep, a pecking cycle is required to clear chips out of the flutes and avoid breaking the drill, and pecking takes time. To eliminate that "continual up and down dance," OSG Tap & Die Inc., Glendale Heights, Ill., developed FTO-GDXL deep-hole, through-coolant drills, which have a special flute form design and coating to enhance chip evacuation, according to Roger Goble, the toolmaker's West area manager. Instead of taking minutes to drill a hole with pecking, he said the OSG tools can drill most deep holes in 5 to 20 seconds. "When I show these deep-hole drilling capabilities to customers, popping holes up to 10" deep in seconds, they just look at me in shock."

Goble added that EDMing and gundrilling are alternative methods for making deep holes, but EDMing takes significantly longer than pecking and requires a specialized machine. In addition, coolant is required to flush chips when gundrilling because a gundrill doesn't have a helix and therefore the tool's straight flute is ineffective in helping to evacuate chips.

Pecking also leaves marks on the ID. "Without pecking, the hole's finish is better," Goble said.



OSG Tap & Die

OSG Tap & Die's FTO-GDXL drills have a special flute form design and coating to enhance chip evacuation so pecking isn't required even when drilling holes more than 4 diameters deep.

Seco Tools Inc., Warren, Mich., also offers drills for making deep holes without pecking. Jay Verellen, manager of the toolmaker's rotating products group, said a through-coolant indexable drill for roughing holes up to 5 diameters deep is the most popular, and replaceable-end or solid-carbide drills are available for more precise holemaking.

The indexable drills, which range from 3/4" to 1 1/4" in diameter, have a relatively open flute gullet to the top of the drill and a nickel-Teflon coating to enhance chip evacuation. "That provides a smooth surface somewhat like a new frying pan," Verellen said.

He concurred that a hole's surface finish is finer without pecking, which is beneficial when reaming is required because each peck can slightly alter the hole's direction and create an inconsistent hole after reaming.

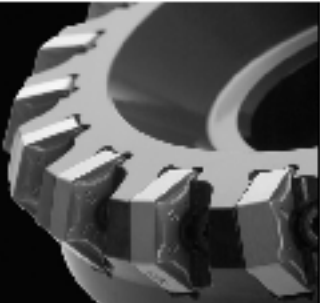
For larger holes, such as a 3"-dia. one, circular interpolation may be a more effective option. "To turn a 3" drill is not so easy for all machines," Verellen said.

Illinois ProTurn, McHenry, Ill., produces components primarily for plastic-injection moldmakers, as



Seco Tools' 217.21 high-feed cutter is for copy and plunge milling.

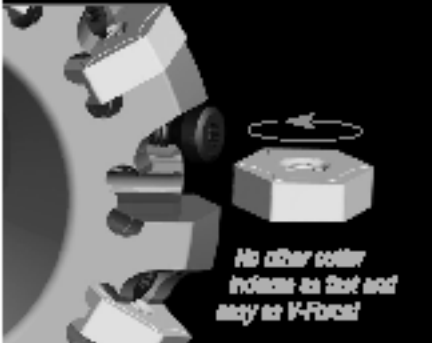
Seco Tools



## VFORCE<sup>™</sup> with SkidLok<sup>™</sup> technology

**The SkidLok Advantage:**


- Insert screw stays in the body
- Faster indexing
- No lat screws
- Single spare part reduces inventory



No other cutter indexes as fast and easy as V-Force!

**SkidLok Pocket Design:**

- Secure insert retention
- Higher insert density
- Increased metal removal rates in cast iron and C&I



**Product Offerings:**

- 2" to 12" diameters
- Coarse and fine pitch
- Saw even more! Available repair inserts let you rough and finish with the same cutter

For more information on V-Force tooling:  
[www.valenite.com/vforce](http://www.valenite.com/vforce)

## Valenite

WE NEVER STOP...

# INTRODUCING THE NEW '08 GS TOOLING CATALOG



## Benefits of GSK Tooling pre-balanced Toolholders:

- Balanced Toolholders at Regular Toolholder price
- CAT40 and BT30/40 Tooling Balanced to 20,000 RPM
- CAT/BT50 Tooling Balanced to 15,000 RPM
- AT3 or better taper accuracy
- Proven Performance of increased tool life

**STOCKED  
IN CANADA  
CALIFORNIA  
CHICAGO**

Visit us at Westec  
Booth #3744  
[www.iscarusa.com](http://www.iscarusa.com)  
1 800 293 8221

## Top Tools *(continued)*

well as for the automotive industry, and drills holes up to 10 diameters deep without pecking, said Dan Lamz, the shop's president. He added that most deep-hole drilling is done in the soft state on a lathe before a workpiece is heat treated, but not exclusively. Drilling hardened material, such as H-13, 4-20 and F-7 steel, is done on a milling machine—and done aggressively. “A ½”-dia. hole could be 2” deep, and we’re drilling it about 1 ipm in 54- to 56-HRC steel,” Lamz said.

### Upping the Feed

In contrast to high-speed machining, where a cutting tool takes large axial cuts with small radial cuts, high-feed machining is defined by small axial cuts with nearly full-diameter radial cuts at high feed rates. That creates a more consistent surface after roughing, so less semifinishing is required, said Verellen, adding that fewer tools are needed. “For a large mold, you may have traditionally roughed it with a 4” tool, remachine it with a 2” and then remachine it again with a 1” and so on,” he said. “With high-feed machining, you tend to use a smaller diameter tool because of the higher velocity, so you may just do the whole mold with a 2” tool and remachine it with a ¾” or smaller tool, reducing overall cycle time.”

Using fewer tools means fewer tool changes and more time in the cut. “The time on the machine is what costs money,” Verellen said. “You can reduce the machine time by reducing the amount of time cutting air and maximize the time cutting chips. That’s where high-feed machining is beneficial.”

High-feed machining maintains high loads, or high feed rates per tooth, and the cutting tools for those applications must be capable of handling those loads. To meet this need, Iscar Metals Inc., Arlington, Texas, offered FeedMill, a high-feed cutting tool that incorporated an insert with a boss on its backside that corresponded with a pocket in the cutter body. This design reduced the force exerted on the screw that holds

the insert in place and enabled the cutter to withstand the high cutting forces created by those loads, said Thomas Raun, the toolmaker's national products manager/die and mold specialist. “The screw is typically a weak point of the cutter body,” he said.

That tool continues to be successfully applied for high-feed machining, but some “design limitations” led Iscar to develop its next generation of high-feed cutting tools—the HeliDo UpFeed line. To improve the design, Iscar incorporated its dovetail clamping method, where an undercut, or taper, in the pocket allows an insert with mating



*Delcam/OSG Tap & Die*

By taking lighter DOCs with smaller diameter tools during roughing (bottom), tools are able to feed at 100 to 200 ipm and accomplish three things: reach into areas that bigger tools cannot, leave smaller stair steps and a more uniform shape that's closer to the final part, and have a cycle time similar to that for semifinishing and finishing.

geometry to dovetail securely into the pocket and reduce the forces on the screw. “It enables running at higher feed rates with predictability and reliability,” Raun said.

To make the insert more economical, the latest insert is pressed to create cutting edges on both sides instead of the previous one-sided version, doubling the



number of edges to six. Iscar's pressing technique also forms top rake geometry on the insert, which reduces spindle load by 20 to 30 percent compared to other high-feed offerings, according to Raun. "It's Z-level roughing on steroids," he said. "You're taking small steps in the Z level but at a very high feed rate. You're probably increasing your metal-removal rate by 50 percent over current methods, and in many cases, it can be well over 100 percent."

Raun noted that the cutter can sustain a feed rate of 0.130 ipt when cutting carbon and tool steels and some free-machining stainless steels and requires a lower feed per tooth when cutting exotics, such as titanium and high-temperature alloys.

For machining larger molds, Goble said OSG offers Tungaloy inserts with a geometry that pushes the cutting forces axially toward the spindle instead of radially to minimize vibration and enable high-feed facemilling at 100 to 300 ipm. "Normally, facemills traverse at 50 ipm if you're lucky," he said.

Goble added that roughing at a high feed enables taking a small DOC, so when machining a cavity level by level, the stair steps, or scalloped height of the material, remaining on the cavity walls are smaller than when hogging and taking a large DOC. This enables faster semifinishing because there is less material to remove. "We want to have a relay race," he said. "We don't want to have the first tool do well and leave the rest of the tools to struggle."

For solid-carbide endmills ½" and smaller, Goble said OSG has tools that can be fed briefly at up to 1,000 ipm in P-20 steel at a light DOC of about 5 percent of the tool's diameter.

#### Don't Burn Those Ribs

A sinker EDM is one option for producing a mold's ribs. That requires



*Iscar Metals*

The Helico Upfeed cutting tool from Iscar Metals accepts inserts that are pressed to create cutting edges on both sides.

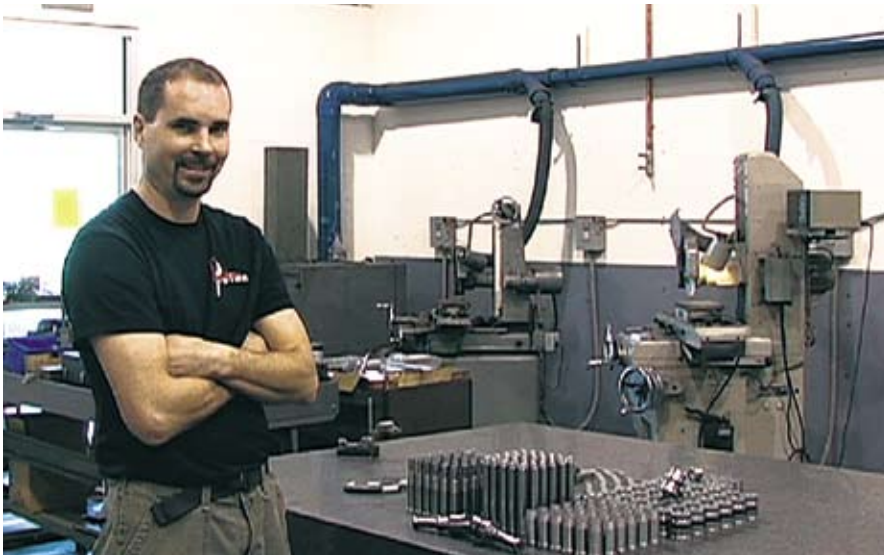
creating a model of the electrode, developing the toolpaths for that model in a CAD/CAM system and then machining the electrode in-house or outsourcing the work. "You went through three processes and haven't cut anything on the mold yet," Goble said. "With rib cut-

ters, you can forget about all that, and machining ribs is up to 20 times faster than burning them."

Because ribs have a draft, or taper, to enable a part to be ejected from the mold, rib cutters are available in widths and with a taper along the shank and

# The Daily Grind

**MONSTER TOOL CO.**  
**PHONE: 888-CARBIDE (227-2433)**  
**FAX: 888-ENDMILL (363-6455)**  
**sales@monstertool.com      www.monstertool.com**



Dennis Spaeth

Dan Lamz founded Illinois ProTurn after working in the industry 20 years.

## Living the dream

**LIKE RESTAURANTS OPENED BY** chefs who want to take their careers to the next level, machine shops are often established by machinists who have chips and coolant coursing through veins and want to work for themselves. Dan Lamz is one such machinist. After 20 years in the metalworking industry, he founded Illinois ProTurn in 2005 to make components for plastic-injection moldmakers.

With a partner whom he's since bought out, Lamz began operations in Cary, Ill., with a Mazak Nexus 200 turning center and now has three. "Within 6 months, we hired a full-time guy, bought another lathe and another 6 months down the road bought two CNC mills," he said. "About 6 months ago, we bought another lathe."

Eventually, space got tight and he moved to a larger facility in McHenry, Ill. There, business has continued to grow and ProTurn is in the process of moving to another location in Cary with double the space to 4,000 sq. ft. The additional space will allow ProTurn to add another lathe and machines for DedTru grinding, centerless grinding and gundrilling.

To be successful, ProTurn keeps its delivery times short and its prices lower than the competition. "We turn around jobs for 30-piece tools with five or six different components from start to finish in 2 weeks," Lamz said. "I don't think I've ever quoted more than 4 weeks no matter how big the package was."

In addition to working 55 to 80

hours a week, ProTurn is able to deliver parts quickly by hard turning and hard machining instead of performing more time-consuming grinding and EDMing operations on hardened workpieces. Lamz attributes his ability to hold tight tolerances when hard turning to his previous side-by-side experience of OD grinding and hard turning. "You use the same techniques on a lathe that you do on a grinder to assure a part is concentric and square," he said.

ProTurn tries to maintain a 2-week backlog. "When I get a 1-week backlog, I start to worry about where my next job is coming from," Lamz said, "and when I'm up to a 3-week backlog I'm wondering how I'll get it all done."

For now, the work flow suits him. "We never get too overwhelmed where we're missing our deliveries, and we never seem to run out of work," Lamz said. "We had tremendous growth last year and have been very fortunate."

But experiencing that growth doesn't mean he's looking to become a large company—a desire probably all entrepreneurs have at least in the back their minds when starting. "It's very easy to control at this level," Lamz said. "It's not a lot of worries but enough. If I could have thought of the best-case scenario 3 years ago, I don't know if it would have surpassed where I am. This has been a dream come true."

—A. Richter

## Top Tools *(continued)*

flutes to match the ribs. "The rib cutter is a male version of the rib being cut," Goble said.

Iscar's Raun said the toolmaker offers a line of solid-carbide mills for rib processing operations. The line includes ballnose, 90°, taper end and taper ballnose mills from 0.4mm to 12.0mm in diameter with shallow, short flutes for creating a thick core diameter and adding stiffness, and a neck diameter with relief from the flute to provide clearance between the cutting edge and the tool's neck.

Tri-Star Mold Inc., Ontario, Canada, specializes in smaller multicavity molds

*PCBN's hardness—second only to diamond—enables an end user to achieve a high level of repeatability when machining multiple features.*

primarily for the automotive and medical industries. Tri-Star's rib machining requirements are somewhat unique, said Dave Tomic, the company's CNC manager. "Some of our parts demand odd-sized ribs," he said. "These ribs are deep and need to be accurate, with fine surface finishes so standard-size rib cutters won't work. We have to profile them with long pencil-neck ballnose or bull nose cutters."

Tri-Star machines to tolerances as tight as 5 to 10 microns and therefore needs cutting tools produced and certified to equally tight tolerances. "Normally, everything we use is within 10 microns or less," Tomic said.

Nonetheless, moldmakers might require EDMing of deep ribs with intricate details, such as a small corner radius, but that doesn't necessarily mean burning the entire feature. "You machine as much as possible and reduce the EDMing requirements," said Seco's Verellen. "Instead of sinking in a three-

step process, you may be able to touch off with one or two steps. It reduces the total time. Whenever possible, we try to eliminate the EDM step by using high-precision tooling, which can be as small as 0.008" in diameter."

And that's something moldmakers certainly desire. "Moldmakers are looking to save as much lead time as they can, and they found that cutting ribs into hard material ends up saving a lot of money in the long run," said Single Source's Kline. "And they get an excellent finish."

### Targeting Tight

As previously mentioned, if the cutting tool is not precise, the part can't be precise. One method for ensuring higher accuracy tools is applying ground inserts instead of molded ones, according to Jeff Hesse, president of Tool Fabrication Corp., Milwaukee. "When you swap a dull insert for a sharp insert, the repeatability from insert to insert is going to be closer if you're using a precision ground insert," he said. "The molded insert is going to have more variation on the edge and the screw hole location. As a result, you're going to have more play in what the cutting tool does."

The tools can be made using a carbide shank with a steel head brazed onto it, and the head has a machined slot to accept the blade, or insert. "You have a replaceable-blade tool that has the rigidity of a solid-carbide tool," Hesse said. The blade is held in place by a screw that goes from outside the slot, through the

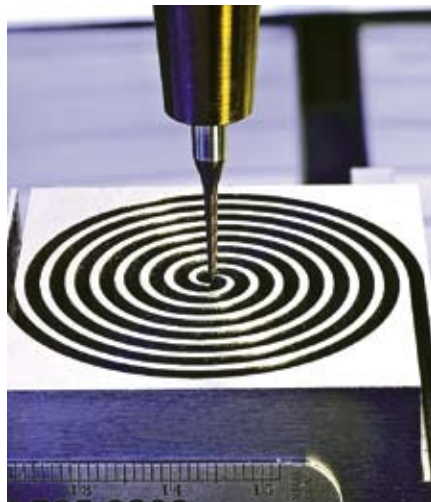
blade and the slot's backside. "That cutting edge is perfectly aligned in the tool for minimum runout," Hesse added.

Carbide shanks are standard, and specials with heavy-metal shanks can be ordered. Carbide is more rigid but the costlier heavy metal can provide "some-what of a dampening effect," Hesse said. "There's some experimentation going on here with regard to utilizing heavy-metal shanks, but heavy metal is a very expensive material, and the cost

vs. benefit may be marginal."


To ensure that OSG high-accuracy ballnose endmills are in specification, some with a tolerance of 3 microns, Goble said the toolmaker uses special machines to grind the tools and inspects each one. "We check them every 10° on the radius, and we provide a certification report in the box with the tool," he said.

Single Source Technologies also provides certification that the solid-carbide




Single Source Technologies


An NS Tool MRT-425 series 4-flute taper endmill cuts deep spiral ribs.




## HIGH PERFORMANCE MILLING




**MWH Series**  
For high production milling




**Diamond Coated Series**  
For graphite milling




**GNG Series**  
For high production milling




**CCMG45 Series**  
For hard milling



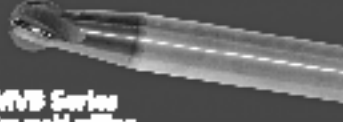
**ALMG Series**  
For aluminum milling




**TM-H Series**  
For thread milling




**ELMG Series**  
For aluminum milling




**MVE Series**  
For mold milling



**EMG3 Series**  
For high temp alloy milling



**Wolver Chuck Series**  
For high performance holding



**EMG6 Series**  
For high temp alloy milling

End mills, drills, countersinks, and much more  
Extensive line of M42 cobalt and mills also available

**MELIN TOOL COMPANY**  
5545 Venture Drive • Cleveland, Ohio 44178  
800-521-1078 • Fax 330-521-1538

[www.endmill.com](http://www.endmill.com)

## Top Tools (continued)

round tools from NS Tool it distributes have radii within 5 microns. In addition, Single Source offers the "extremely accurate" Future Generation Tools line from Pokolm-Voha for machining steel, aluminum and materials hardened up to 64 HRC, according to Kline.

In addition to producing accurate parts, high-precision cutters, such as

matched sets of endmills that are within 0.0002" from one to the next, reduce programming time. "If you program for a 0.250" endmill in CAD/CAM and one endmill measures 0.247" and the next 0.249", and you're trying to hold a couple tenths tolerance on a workpiece, you're never going to achieve that when the cutters you have aren't what you thought they were," said Iscar's Raun.

"You save yourself a lot of extra work by pursuing and having companies supply you with matched sets of endmills."

On the other hand, applying certified tools isn't absolutely necessary for achieving tight tolerances. ProTurn's typical tolerance is 0.0002"/-0.0000", and Lamz said the shop primarily uses standard, off-the-shelf tools. "We specialize in close-tolerance work that most people would grind," he said. "Rather than grinding and burning, we are hard turning and hard machining."

### Hard for Hard

One of the exceptions to ProTurn's use of standard tools is a special PCBN-brazed insert for hard turning. The insert starts as a Kennametal carbide insert and once worn is then sent to Sumitomo Electric Carbide. Sumitomo then grinds a pocket into the insert, brazes in a PCBN tip and hones the cutting edge. "I'm reusing the insert to make a new tool," Lamz said.

**Hard Metal? Graphite?**

**ROBB JACK CORPORATION**  
Manufacturers of Premium Rotary Cutting Tools

**1-800-527-8888**  
[www.robbjack.com](http://www.robbjack.com)  
3300 Wisconsin Road, Lincoln, CA 95804  
Phone: 916 645-8045 Fax: 916 645-8106

**Crystallume**  
Engineered Diamond Products  
A Division of RobbJack Corporation

**1-800-789-4322**  
[www.crystallume.com](http://www.crystallume.com)  
8307 De La Cruz Blvd., Santa Clara, CA 95054  
Phone: 408 654-1700 Fax: 408 654-1710



Finish machining of a mold cavity insert with a PCBN microtool.

Tri-Star Mold

Because PCBN is not a low-cost cutting tool material, ProTurn avoids applying them for interrupted cuts to prevent breakage and premature wear. To achieve that, the shop doesn't perform soft machining. Instead, workpieces are turned in the soft state, leaving about 0.005" of stock per side, heat treated and hard turned. "Then we throw them one time into a machining center and do everything hard," Lamz said. "If I have interrupted cuts, I can't hard turn it." Of course, exceptions exist, and when he can't avoid hard turning with interrupted cuts, Lamz effectively applies a Kennametal CCMT 3251 carbide finishing insert that ProTurn "stumbled upon by accident," he said. "The insert's not even designed for that."

Besides being effective for hard turning, PCBN's hardness—second only to diamond—enables an end user to achieve a high level of repeatability when machining multiple features. "Say

you're doing a five-cavity mold and you need the fifth cavity to be as accurate as the first four. CBN tools let you accomplish that because there's hardly any tool wear," said OSG's Goble.

Lamz said the wear rate for PCBN is "similar to a bell curve," with the material wearing about 0.0001" after machining three or four pieces, then stabilizing and sustaining the cutting edge for a long time before wearing again at the end of its life.

The tool material's long and predictable life make PCBN appropriate for unattended machining, primarily finishing, while imparting fine surface finishes because the less a tool wears, the less material is left on a part. "With CBN, you take lighter chip loads but at higher surface parameters, so the finishes become finer. That's how you reduce benching and polishing," Tri-Star's Tomic said. "We do a lot of lights-out machining, and if we can run lights-out

## contributors

**Illinois ProTurn**  
(847) 341-6662  
www.ilprotorn.com

**Iscar Metals Inc.**  
(817) 258-3200  
www.iscarmetals.com

**OSG Tap & Die Inc.**  
(630) 790-1400  
www.osgtool.com

**Seco Tools Inc.**  
(586) 497-5564  
www.secotools.com

**Single Source Technologies Inc.**  
(248) 232-6232  
www.singlesourcetech.com

**Tool Fabrication Corp.**  
(800) 790-8665  
www.toolfab.com

**Tri-Star Mold Inc.**  
(519) 624-1991  
www.tri-starmold.com

The **Click Change™** Insert Holder

**Five Second Insert Changes**

- No lost wrenches • No stripped screws • .0002" repeatability • ISO/ANSI standard CNMG, VNMG, VMG, & more • No adjustments • No wear components • U.S. Patent 7163069 • Foreign patents pending •

**Axian Technology 800-456-3131**  
**www.clickchange.com**

## Top Tools (continued)

and guarantee high accuracy, it's easy to justify."

So although PCBN tools command a premium price, reducing the amount of labor to manually finish a mold provides a quick return on investment. "I've got a customer that was able to reduce benching time from 22 hours to 5 hours using CBN tools," said Single Source's Kline. "When you're talking a burden rate of \$70 to \$125 in most shops, do the math and you're saving a significant amount of money."

To maximize PCBN's benefits, PCBN tools need to be applied on a "technologically advanced machine," Kline added. "If someone has an older machine with an 8,000-rpm spindle, CBN is not the tooling I want to move him into. He's just going to get upset."

[www.ctemag.com](http://www.ctemag.com)

For more information, visit "Articles Archive Index" and enter the keyword "moldmaking."

## keywords

**FEED:** Rate of change of position of the tool as a whole, relative to the workpiece while cutting.

**PCBN, POLYCRYSTALLINE CUBIC BORON NITRIDE:** Cutting tool material consisting of polycrystalline cubic boron nitride with a metallic or ceramic binder. PCBN is available either as a tip brazed to a carbide insert carrier or as a solid insert. Primarily used for cutting hardened ferrous alloys.

—CTE Metalworking Glossary

That's because PCBN requires cutting at high feeds to maximize the material's benefits.

Care must also be taken in how PCBN tools, which are harder and therefore more brittle than carbide, are applied, with Iscar's Raun recommending not removing more than 0.005" of stock when finishing. "It's not as forgiving as carbide," he said. "Any mistake when programming or extra material in a workpiece and you can kiss that CBN goodbye."

The tools that cut the metal are critical, but moldmakers must still take into account the machining system to

remain competitive. "Whether it's the machine, toolholders, cutters, programming strategies, the setup, it's the whole combined package that's important," said Tri-Star's Tomic. "You can have the best machine or cutter in the world and if you don't use it in the right package, it gets you nowhere."

CTE

### About the Author:

Alan Richter is editor of *Cutting Tool Engineering*. He can be reached at (847) 714-0175 or via e-mail at [alanr@jwr.com](mailto:alanr@jwr.com).



Visit our website & order your  
**NEW Free Catalog today!**

# XEBEC™

**BRUSHES FOR AUTOMATED  
DEBURRING & POLISHING**

**DEBURRING  
STONES & BRUSHES**

**POLISHING  
STICKS & BOBS**

**SURFACES CROSS-HOLES MOLDS**

**DEBURRING**  
6241 Expansion Way  
Huber Heights, Ohio 45424  
Toll Free 877-254-8300  
[www.deburringtechnologies.com](http://www.deburringtechnologies.com)