# EDGE²™

## REVOLUTION

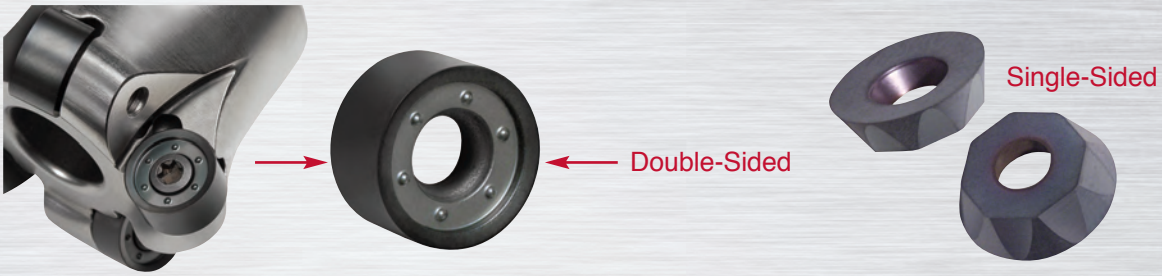
ROUGHING FINISHING

DAPRA®

# DTB Series Double-Sided Inserts and Cutter Bodies: Twice the Metal Removal – Twice the Value

## DTB Series Inserts

**Dapra's *NEW* inserts are double-sided, allowing twice the number of usable edges versus a normal, single-sided insert.**




- Outstanding insert life for roughing applications – significantly outlasts single-sided button inserts, as well as competitors' double-sided offerings
- Thicker carbide insert provides better heat and stress absorption
- Smooth cutting action, with available geometries suitable for cutting virtually all materials (except bar-stock aluminum and plastics)
- Anti-rotation raised dimples for securely and consistently seating insert in cutter body when indexing
- Recessed mounting face creates unobstructed chip flow for reduced stress and heat
- Capable of metal removal rates rivaling high-feed, but with better reliability

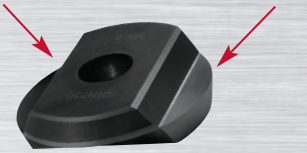
## DTB Cutter Bodies

- Super-tough, high-shock tool steel
- Fully thru- and case-hardened bodies for optimum strength and longevity
- High accuracy/centrality for minimal runout, providing best finishes and tool life
- Anti-rotation recessed dimples correspond with raised insert dimples for secure and consistent seating and indexing
- Thru-coolant standard in many bodies for optimum air blast or high-pressure coolant use

## EDGE²™ DTB Roughing Tools Index

# Dapra's New Spherical Ball Nose System: Truly Indexable Finishing Inserts!

## 2x 2 usable cutting edges (not just 1!)




### The SBN System features:

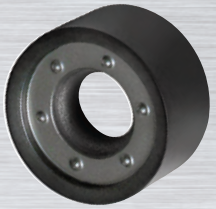
- Cost savings – get two edges instead of just one
- Thicker carbide insert construction for better heat and stress absorption
- Superior surface finishes due to an improved grinding process
- Outstanding insert life for finishing applications – allowing the longest unattended runs
- Excellent tip-cutting or 3D profiling
- Fully thru-hardened bodies for optimum strength and longevity
- High accuracy/centrality for minimal runout, providing best finishes and tool life
- Close-tolerance insert pocket produced by EDM, removing less body material for improved insert support

## EDGE²™ SBN Finishing Tools Index

# DTB Roughing Series

EDGE²™

## Button Inserts



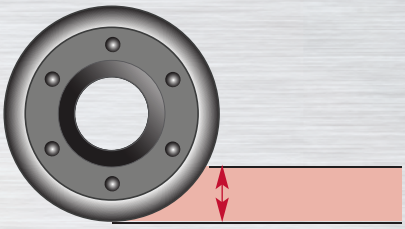
***Round inserts have a strong cutting edge, making them an excellent choice for many applications, including:***

- Mold cavity and core roughing
- Roughing of complex part contours
- Helical interpolation (larger diameter hole making)
- Pocketing
- Semifinishing
- Face milling

*"T" inserts have a reinforced cutting edge for steels, irons and hard milling.*

*"D" inserts have a positive cutting geometry, designed to reduce cutting forces and heat. This edge is suitable for all materials.*

| Insert Part Number | IC | Thick-ness | # of Usable Edges | Radius | Edge Type |
|---|---|---|---|---|---|
| DTB-12-T | 12mm | .250" | 12 | 6mm | Strongest |
| DTB-12-D | 12mm | .250" | 12 | 6mm | High-Shear |



***0.100" Max. DOC for 6 Indexes Per Side***

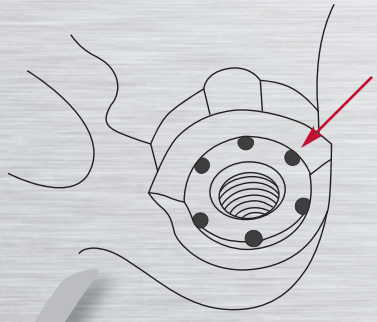
### *Grade Availability**

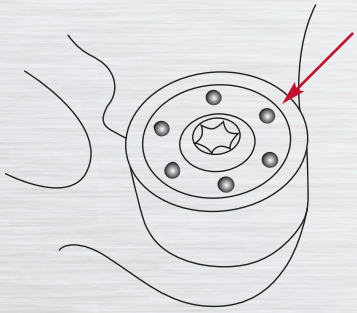
| | Uncoated | with Coating | |
|---|---|---|---|
| TOUGHEST Shock Resistance | DMK30 T, D | DMK30-TCI T, D | DMK30-GLH T, D |
| MEDIUM Shock and Wear | DMP25 T | DMP25-TCI T | DMP25-GLH T |
| HARDEST Wear Resistance | DMK15 T | DMK15-TCI T | DMK15-GLH T |

*** See page 9 for insert grade descriptions.***

## Loading

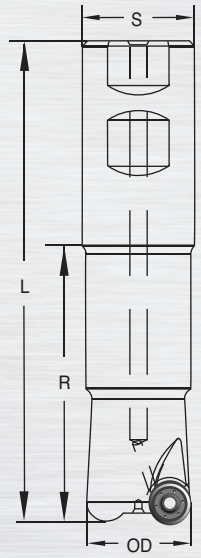
Your DTB series tool has hemispherical dimples designed to provide anti-rotation support as well as help maintain a minimum of 6 indexes per insert side (12 indexes total per insert). It is important that the insert be loaded correctly – with the raised dimples on the insert seating down into the recessed dimples in the cutter body pocket. See the diagram below.




***See chart on page 8 for technical help on optimum feeds for button inserts.***
***See page 9 for insert grade descriptions.***
***See page 10 for recommended speeds/feeds.***

## Cutter Bodies

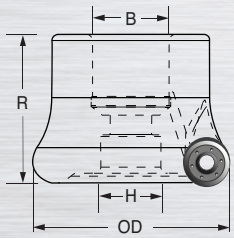



### End Mills (with thru-coolant)

| OD Diameter | Holder | R Effective Length | L Overall Length | S Shank Dia. | Flutes | Insert Dia. |
|---|---|---|---|---|---|---|
| 1.250" | DTEM125-300-R4-3 | 3.00" | 5.40" | 1.25" | 3 | 12mm |
| 1.500" | DTEM150-350-R4-3 | 3.50" | 5.90" | 1.25" | 3 | 12mm |

### Carbide Core, Long-Reach End Mills (solid, no thru-coolant)

| OD Diameter | Holder | R Effective Length | L Overall Length | S Shank Dia. | Flutes | Insert Dia. |
|---|---|---|---|---|---|---|
| 1.250" | CC-DTEM125-500-R4-3 | 5.00" | 7.40" | 1.25" | 3 | 12mm |
| 1.500" | CC-DTEM150-550-R4-3 | 5.50" | 7.90" | 1.25" | 3 | 12mm |

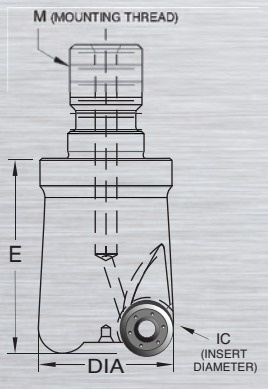



### Shell Mills

| OD Diameter | Holder | R Effective Length | B Arbor Dia. | H Counter Bore Dia. | Flutes | Insert Dia. |
|---|---|---|---|---|---|---|
| 2.000" | DTSM200-075-R4-5 | 1.50" | 0.75" | 0.59" | 5 | 12mm |
| 2.500" | DTSM250-100-R4-6 | 1.75" | 1.00" | 0.79" | 6 | 12mm |
| 3.000" | DTSM300-100-R4-7 | 2.00" | 1.00" | 0.79" | 7 | 12mm |
| 4.000" | DTSM400-150-R4-8 | 2.00" | 1.50" | 1.10" | 8 | 12mm |

*See page **22-23** for Modular Extensions.*

## Screw-On Modular Heads




### *Screw-On Heads Fit Industry Standard Cutting Systems*

- Compatible with ISO standard modular systems
- Greater effective reach than solid end mills
- Close-tolerance mounting minimizes runout
- Standard inch wrench flats – no special metric wrenches needed

| Dia. | Holder | M | E | Insert IC | Flutes | Wrench |
|---|---|---|---|---|---|---|
| **Screw-On Heads** | | | | | | |
| 1.250" | DTEM125-MOD-R4-2 | M16 | 1.75" | 12mm | 2 | 15/16" |
| 1.500" | DTEM150-MOD-R4-2 | M16 | 1.75" | 12mm | 2 | 15/16" |

# DTB Application Information EDGE²™

## Recommendations


- Tool is most appropriate for "Z-level" roughing; ramp to Depth of Cut (DOC) and clear entire level.
- Plunging is not recommended with DTB series cutters. Ramping (up to 2° max.) is recommended.
- Try to maintain at least 75% of the cutter diameter on the workpiece whenever possible. Hanging a cutter off to the side of your work is detrimental to tool life and performance.
- Width of Cut (WOC) should be 60-75% of cutter diameter whenever possible, creating a "scalloping" effect (end mills only) between passes, especially with longer-length tools.
- Round inserts provide a very strong cutting edge and the ability to machine much closer to finish size. Utilize high feeds with light DOC to take advantage of these benefits. High metal removal rates will be achieved without high horsepower consumption.
- Use the Feed Rate Compensation chart on page 7 to compensate for chip thinning that occurs with round inserts. This will provide for optimum metal removal rates and tool life; the lighter the DOC, the more critical feed compensation becomes.

## Technical Considerations

- Always use anti-seize compound on screws.
- Change insert screw every 10 inserts.
- Use the shortest-length tool holder (end mill holder) for maximum rigidity; the shank of the cutting tool should be up inside the machine spindle taper whenever possible.
- Thoroughly clean pocket and screw at each insert change.
- Use tool holders appropriate for roughing operations: end mill holders and power chucks *are* recommended; collets *are not* recommended.


## Spare Parts & Tools

***New cutter bodies may require additional torque to fully seat the inserts. Once the new cutter's pockets are "broken in," the recommended torque specs in the chart can be followed regularly.***

***All tools use Anti-Seize Grease ASG-120***

## Safety

*Modern metal cutting techniques involve the potential use of very high operating parameters (speeds, feeds, depths of cut, etc.). This creates the potential for flying chips and debris, and can also create tool breakage due to a variety of causes. As such, any metal cutting operation should be executed in a completely enclosed (shielded) environment to protect against injury from flying objects. Dapra does not assume responsibility for any loss, damage or expense incurred in any use or handling of our products after purchase.*

*Grinding produces hazardous dust. To avoid adverse health effects, use adequate ventilation and read material safety data sheet first.*

## Running

Your double-sided DTB series inserts are very thick, providing outstanding strength and superior tool life in operation. With this increased thickness comes the requirement that the feed per tooth (FPT) not exceed the clearance the insert allows. In general, it is recommended to initially not exceed an FPT of .025". Under certain circumstances (lighter DOC and/or WOC) the feed can be higher, but the potential for rubbing (or heeling) of the back-side cutting edge exists. When trying heavier feeds, evaluate the back-side cutting edge after running the tool for only a few minutes, looking for evidence of rubbing or damage.

Your DTB series inserts utilize chip-thinning for optimum performance. Chip-thinning is the process in which the actual chip thickness created by a cut is less than the FPT at which the tool is programmed. See the chart below to see the chip-thinning factor for various depths of cut. *For a correct chip thickness, multiply the FPT (from page 10) by the chip thinning factor below.* This will give you an accurate feed and optimize your performance.

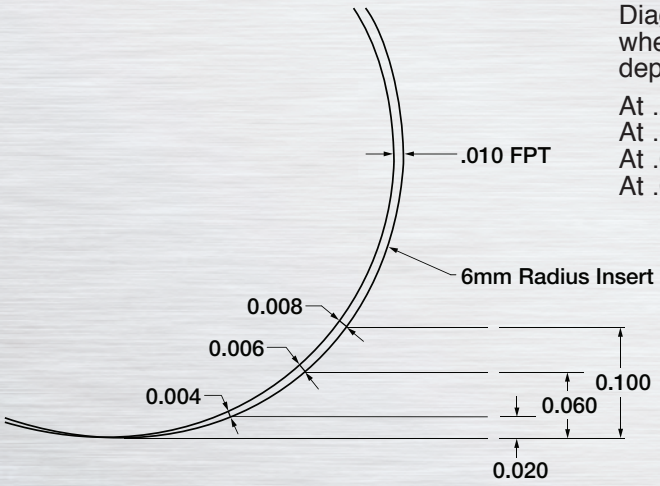



Diagram (left) shows the varying chip thickness when feeding .010" feed per tooth (FPT) at different depths of cut (DOC) with a 12mm diameter insert:

At .250" DOC, the chip thickness is a true .010"
At .100" DOC, the chip thickness is .008"
At .060" DOC, the chip thickness is .006"
At .020" DOC, the chip thickness is only .004"

## Feed Rate Compensation

After determining the desired chip thickness (FPT – see chart on page 10), find the Depth of Cut intersection in the chart at right. Multiply the desired chip thickness by the factor shown in the chart. This will be the Adjusted Feed per Tooth (AFPT), resulting in a true chip thickness of the desired amount.

**Example:**

@ .03" Depth of Cut (DOC), the factor for the chip thickness = 2.1.
So, if a chip thickness of .005" is desired, a feed rate of .0105" (.005 x 2.1) needs to be programmed into the machine tool.

***or***

Adjusted Feed per Tooth (AFPT) = chip thickness x chip thinning factor (from chart)

| Depth of Cut (DOC) | Insert Diameter 12mm |
|---|---|
| 0.005 | 5 |
| 0.010 | 3.6 |
| 0.015 | 2.9 |
| 0.020 | 2.6 |
| 0.025 | 2.3 |
| 0.030 | 2.1 |
| 0.035 | 2 |
| 0.040 | 1.8 |
| 0.050 | 1.7 |
| 0.060 | 1.5 |
| 0.075 | 1.4 |
| 0.085 | 1.3 |
| 0.100 | 1.3 |
| 0.125 | 1.2 |
| 0.150 | 1.1 |
| 0.180 | NR |
| 0.200 | NR |
| >0.20 | NR |

# DTB Troubleshooting & Helical Interpolation

EDGE²™

## Troubleshooting

| Concern | Possible Cause | Solutions |
|---|---|---|
| **Insert wear appears high (flank wear)** | • Not enough chip load<br>• Surface footage is high<br>• Incorrect grade or coating | ❑ Verify correct speed and feed<br>❑ Increase feed rate<br>❑ Decrease RPM<br>❑ Increase DOC<br>❑ Use harder grade |
| **Insert chipping** | • Surface footage is low<br>• Incorrect grade or coating<br>• Using Dished insert incorrectly<br>• Feed too high | ❑ Verify correct speed and feed<br>❑ Increase spindle speed<br>❑ Decrease feed rate<br>❑ Decrease DOC<br>❑ Use T-Land Insert<br>❑ Use tougher grade |
| **Built-up edge on insert** | • Low surface footage<br>• Light chip load (feed per tooth)<br>• Incorrect coating | ❑ Verify correct speed and feed<br>❑ Increase cutting speed<br>❑ Increase feed rate<br>❑ Select different coating<br>❑ Use coolant |
| **Poor finish/chatter** | • Cutter hung out too far<br>• Excessive runout<br>• Inadequate tool holding | ❑ Reduce tool gage length<br>❑ Check tool holder wear<br>❑ Use high-rigidity tool holder |
| **Tool shank breaks** | • Tool pressure too great<br>• Fatigued cutter body | ❑ Decrease DOC<br>❑ Reduce tool gage length<br>❑ Decrease feed rate |

## Hole Diameter Calculation

| Shell Mill Part Number | Minimum Hole Dia. | Maximum Hole Dia.* |
|---|---|---|
| DTSM200-075-R4-5 | 3.25" | 4.00" |
| DTSM250-100-R4-6 | 4.25" | 5.00" |
| DTSM300-100-R4-7 | 5.25" | 6.00" |
| DTSM400-150-R4-8 | 7.25" | 8.00" |

***Formulas:***

***Minimum Hole Dia.:***

**(Tool Dia. x 2) - (1.5 x Insert Dia.)**

***Maximum Hole Dia.*:***

**Tool Dia. x 2**

** Not generally recommended. At this diameter, the center tip is at its maximum. It is suggested that you stay slightly under this number.*

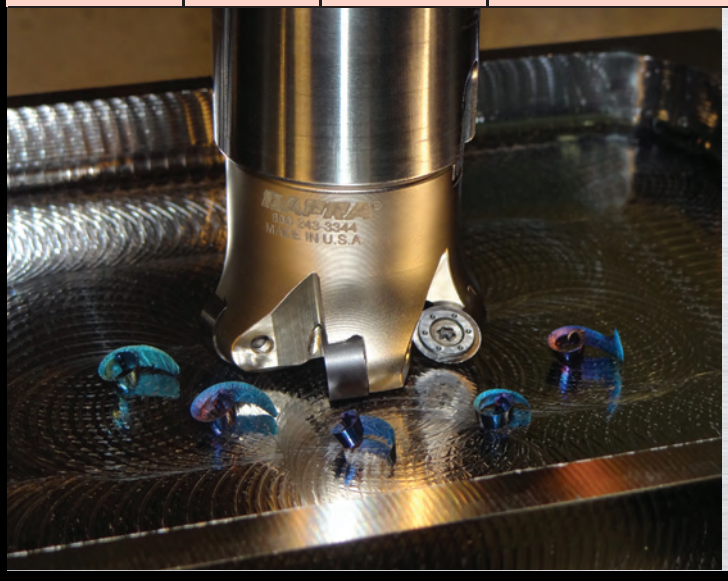
Larger diameter hole making can be quick and easy when a DTB Series Cutter is used in combination with helical interpolation. This technique resembles thread milling in that all three axes (X, Y and Z) are in motion simultaneously. It differs from thread milling in that the tool is introduced into the material without a start hole of any kind. The tool simply is positioned at the inside diameter of the hole to begin its helix from there, achieving complete material removal from the hole by ramping down to the final depth. This smooth operation tends to avoid the high horsepower consumption characteristic of large diameter hole making. This quick and easy process offers the added advantage of allowing many different hole sizes to be generated with the same diameter tool. Hole size variation is all in the programming.

***For more information on how helical interpolation can improve your manufacturing efficiency, contact your Dapra Applications Specialist.***

# DTB Insert Grades

EDGE²™

| Shock & Wear Resistance | Uncoated (Base Grade) | with Coating | Description | Specifications |
|---|---|---|---|---|
| TOUGHEST Shock Resistance | DMK30 | | Moderate wear resistance/high shock resistance. Recommended for interrupted or unstable steel, most stainless steel, high-temperature alloys and cast iron applications. | ANSI C1-C2 ISO K25-K40, M25-M35 |
| | | DMK30-TCI | High-performance medium- to high-temperature grade. Outstanding shock and wear resistance in steels, irons and stainless steels. Best suited for materials < 44 Rc. | |
| | | DMK30-GLH | Premium high-temperature coating. Best resistance to heat for high-shock applications. Excellent for tough stainless steels, high-temperature alloys and many tool steels. | |
| MEDIUM Shock and Wear | DMP25 | | High wear resistance/moderate shock resistance. Recommended for most steel and ductile iron applications. | ANSI C5-C6 ISO P25-P40 |
| | | DMP25-TCI | High-performance medium- to high-temperature grade. Outstanding wear resistance in steels and ductile. Best suited for materials < 44 Rc. | |
| | | DMP25-GLH | Premium high-temperature grade. Unbeatable performance and wear resistance in high-heat applications such as higher-speed machining in steels (< 44 Rc) and ductile irons. | |
| HARDEST Wear Resistance | DMK15 | | Highest wear resistance with reduced shock absorption capabilities. Micro-grain carbide provides excellent edge strength. Suitable for all materials under stable conditions. | ANSI C2-C3 ISO K15-K25, M15-M25 |
| | | DMK15-TCI | High-performance medium- to high-temperature grade. Great for higher-speed gray iron applications and lighter cuts in steels <52 Rc or ductile iron. | |
| | | DMK15-GLH | Premium high-temperature grade for optimum wear resistance in cast irons and steel hard milling > 44 Rc. | |



*Other coatings available upon request*

EDGE² ROUGHING

# DTB Cutting Speeds

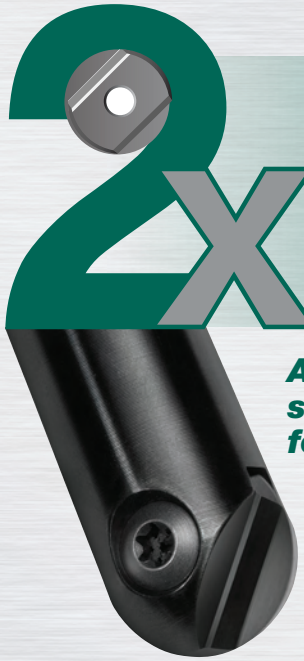
EDGE²™

| Recommended Cutting Speeds for Dapra DTB Cutters | | | 1018, 12L14, 1041, 1045 LOW-TO-MEDIUM CARBON STEELS | 4140, 4150 4340, H13, P20, A2, D2 TOOL STEELS, HIGH-ALLOY STEELS (SOFT) | 4140, 4150 4340, H13, P20, A2, D2 TOOL STEELS, HIGH-ALLOY STEELS (HARDENED) | 303, 304 LOW 400 SERIES FREE MACHINING STAINLESS | 316, 347, PH STAINLESS TOUGHER STAINLESS | GRAY, MALLEABLE, DUCTILE CAST IRONS | 6061, 7075 ALUMINUM ALLOYS | AMPCO, WEARITE COPPER ALLOYS | INCONEL, WASPALOY, MONEL HIGH-TEMP. ALLOYS/ TITANIUM | PLASTICS, NON-FERROUS |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| TOUGHEST Shock Resistance | LOWER TEMPS ▲ HIGHER TEMPS ▼ | DMK30-TCI | 350-550 | 300-500 | NR | **250-500** | **250-450** | 350-600 | NR | 300-800 | **50-200** | NR |
| | | DMK30-GLH | 450-700 | 350-600 | NR | **300-600** | **250-500** | 400-750 | NR | 400-850 | **50-200** | NR |
| MEDIUM Shock & Wear | LOWER TEMPS ▲ HIGHER TEMPS ▼ | DMP25-TCI | **400-700** | **350-600** | NR | NR | NR | **300-650 DUCTILE** | NR | NR | NR | NR |
| | | DMP25-GLH | **450-800** | **400-700** | 200-500 | NR | NR | **300-750 DUCTILE** | NR | NR | NR | NR |
| HARDEST Wear Resistance | LOWER TEMPS ▲ HIGHER TEMPS ▼ | DMK15-TCI | 500-800 | 450-750 | **< 52 Rc 250-450** | 300-600 | 250-650 | **300-750 GRAY** | NR | **400-850** | 50-200 | NR |
| | | DMK15-GLH | 550-900 | 450-800 | **> 44 Rc 250-450** | 300-750 | 250-700 | **400-800 GRAY** | NR | **450-900** | 50-200 | NR |
| 1st CHOICE GEOMETRY | | | **T** | **T** | **T** | **D** | **D** | **T** | **NR** | **D** | **D** | **NR** |
| RECOMMENDED FPT RANGE | | | .005-.015" | .005-.012" | .002-.008" | .004-.012" | .003-.010" | .005-.015" | | .005-.020" | .003-.007" | |

** First choice grade shown in bold text.

The parameters provided are suggested operating parameters. Actual speeds and feeds will depend on many variables, such as rigidity, workpiece hardness, tool extension, machine accuracy, Depth of Cut, etc. Start at the middle of the SFM range and the low end of the FPT range. Next, increase FPT to optimize productivity and tool life. Higher SFM will provide higher output but may reduce tool life. Try different combinations to find the parameters that best suit your needs.

- The -TCI coatings are best suited for low-to-medium operating speeds (temperatures) and softer materials.
- The -GLH coatings are best suited for high operating speeds (temperatures) and harder materials.




## ONLY with Dapra's New SBN tool system can you achieve high-quality finishes AND use a Ball Nose insert twice!

***Achieving this cost savings requires adhering to a few simple application guidelines (see Diagram D1 below for examples):***

**MOST IMPORTANT!** – *Your SBN insert has a full 360 degrees of metal-cutting surface, but it must be used only 180 degrees at a time. All cutting must be kept at or below the insert center line to allow the cutting edges to perform correctly. Any attempts at cutting above the insert center line may result in failure, including insert and/or cutter damage.*

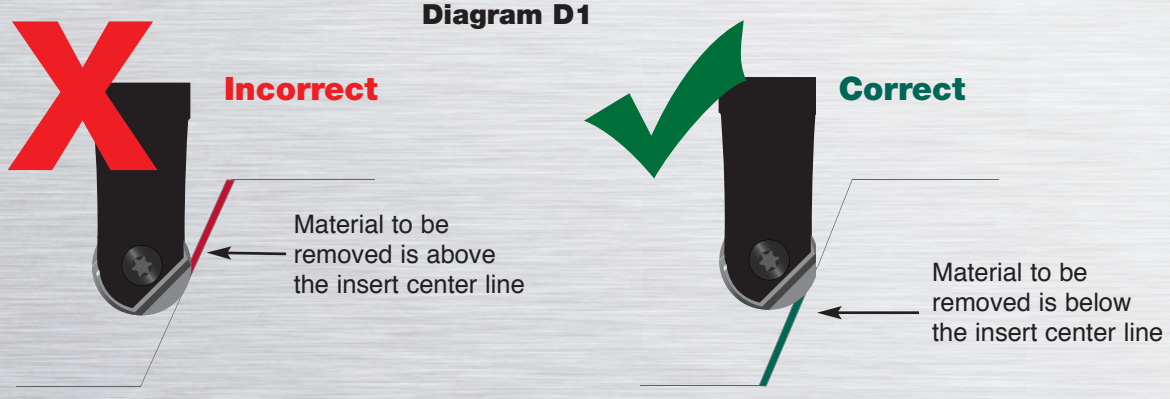
**Finishing –** Use your SBN insert for finishing, preferably with a top-down program. In other words, start your cutting at the top of a part and profile around, slowly working your way down to the bottom (water-line programming/Z-level finishing). A raster-pass (lacing) cut may be used, but take care to leave less than .010" finish stock to avoid cutting above the insert center line. Best performance is achieved in tapered-wall work, but straight-wall finishing can be accomplished by using **ONLY** the "top-down" approach detailed below (see Diagram D1).

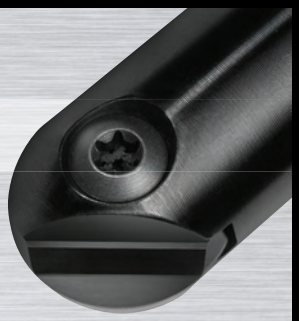
**Semi-Finishing (Light Roughing) –** Use your SBN insert **ONLY** with a top-down approach when semi-finishing or light roughing. Start your cutting at the top of a part and profile around, working your way down. Do **NOT** attempt to lace cut or raster, as rubbing on the back side of your SBN insert may occur (cutting above insert center-line), ruining the surface finish and potentially damaging your SBN insert.

**Roughing –** Your SBN insert is **NOT** suitable for roughing.

***** IN GENERAL, KEEP YOUR CUTTING CONTAINED TO THE BOTTOM 180 DEGREES OF THE INSERT – THIS ALLOWS TWO USES INSTEAD OF ONE!***

**Diagram D1**



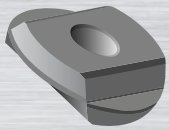


## 1 Step One: Choose Your Inserts

### SPHERICAL BALL NOSE

| Part # | D Ø Diameter | Thickness | Radius | Hole Diameter |
|---|---|---|---|---|
| SBN-0500 | .500" | 0.138" | 0.250" | 0.2070" |
| SBN-0750 | .750" | 0.218" | 0.375" | 0.2070" |
| SBN-1000 | 1.000" | 0.250" | 0.500" | 0.2464" |

- Available grades stocked standard: UC (uncoated) and GLH (coated)



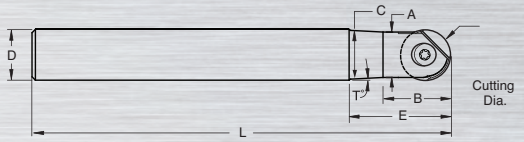
## 2 Step Two: Choose Your Insert Grade

| Uncoated (Base Grade) | with Coating | Description | Specifications |
|---|---|---|---|
| UC | | Micro-grain tungsten carbide with high edge strength and good toughness. Good for machining steels, stainless steels, high-temperature alloys, cast iron and nonferrous materials. | (C-2), (K10) |
| | GLH | Premium high-temperature and high-hardness coating. Optimum performance and wear resistance in virtually all materials makes this an extremely versatile and simple grade choice. | 3600 HV, 2000° F, .2 Co |

# 3 Step Three: Choose Your Cutter Body

## STANDARD SHANK

- *Thru-hardened steel shanks*
- *EDM pocket for optimum strength and accuracy*
- *Straight inch or metric shanks*

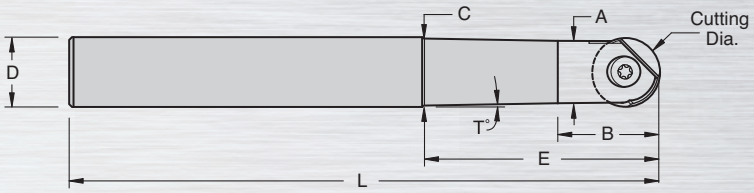



### SBN End Mills – Standard Shank

| | Part Number | Cutting Diameter | A Ø | B Straight Length | C Taper End Ø | D Shank Ø | E Effective Length | T Taper Angle | L Overall Length |
|---|---|---|---|---|---|---|---|---|---|
| Inch Shank | SBNEM-0500-7000-SS | 0.500" | 0.450" | 0.750" | 0.470" | 0.500" | 1.250" | 1.7° | 7.000" |
| | SBNEM-0750-7500-SS | 0.750" | 0.670" | 1.000" | 0.720" | 0.750" | 1.880" | 1.5° | 7.500" |
| | SBNEM-1000-8000-SS | 1.000" | 0.890" | 1.500" | 0.970" | 1.000" | 2.500" | 2.2° | 8.000" |
| Metric Shank | SBNEM-0500-7000-SS-12MM | 0.500" | 0.450" | 1.250" | Straight | 12mm | 1.250" | N/A | 7.000" |
| | SBNEM-0750-10000-SS-18MM | 0.750" | 0.670" | 1.880" | Straight | 18mm | 1.880" | N/A | 10.000" |
| | SBNEM-1000-10000-SS-25MM | 1.000" | 0.890" | 1.500" | 0.970" | 25mm | 2.500" | 1.7° | 10.000" |

# 3 Step Three: Choose Your Cutter Body

## SOLID CARBIDE




***Achieve Maximum Performance with Solid Carbide Cutter Bodies!***

Optimize performance with Carbide Shank tooling:

• ***reduced deflection*** • ***increased stiffness*** • ***less chatter***

• ***heat shrink toolholding capability*** *(SBN with Solid Carbide Shank only)*

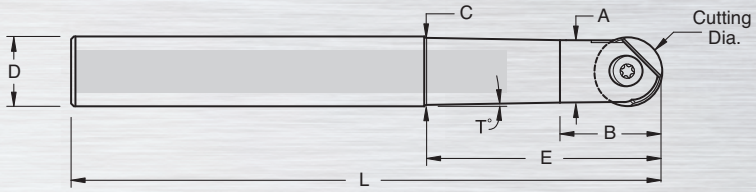
**** Keep brazed joint a minimum of 2" away from heat shrink toolholder.***
***SC (Solid Carbide Shank) tooling is suitable for FINISHING APPLICATIONS ONLY.***
***SC tooling is NOT suitable for roughing and applications with significant heat.***

### SBN End Mills – Solid Carbide Shank

| | Part Number | Cutting Diameter | A Ø | B Straight Length | C Taper End Ø | D Shank Ø | E Effective Length | T Taper Angle | L Overall Length |
|---|---|---|---|---|---|---|---|---|---|
| Inch Shank | SC-SBNEM-0500-7000-SS | 0.500" | 0.450" | 0.750" | 0.470" | 0.500" | 2.000" | 0.70° | 7.000" |
| | SC-SBNEM-0750-7500-SS | 0.750" | 0.670" | 1.000" | 0.720" | 0.750" | 2.500" | 1.00° | 7.500" |
| | SC-SBNEM-0750-10000-SS | 0.750" | 0.670" | 1.000" | 0.720" | 0.750" | 3.750" | 0.50° | 10.000" |
| | SC-SBNEM-1000-8000-SS | 1.000" | 0.890" | 1.500" | 0.970" | 1.000" | 3.000" | 2.25° | 8.000" |
| | SC-SBNEM-1000-11000-SS | 1.000" | 0.890" | 1.500" | 0.970" | 1.000" | 4.500" | 0.75° | 11.000" |
| Metric Shank | SC-SBNEM-0500-7000-SS-12MM | 0.500" | 0.450" | N/A | N/A | 12mm | 1.250" | N/A | 7.000" |

# 3 Step Three: Choose Your Cutter Body

## CARBIDE CORE




***Achieve Higher Performance with Carbide Core Cutter Bodies!***

Optimize performance with Carbide Core tooling:

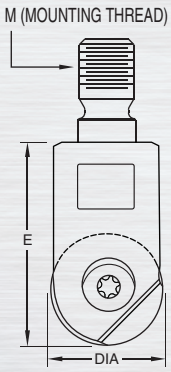
• ***reduced deflection*** • ***increased stiffness*** • ***less chatter***

### SBN End Mills – Carbide Core Shank

| Part Number | Cutting Diamater | A Ø | B Straight Length | C Taper End Ø | D Shank Ø | E Effective Length | T Taper Angle | L Overall Length |
|---|---|---|---|---|---|---|---|---|
| CC-SBNEM-0750-9000-SS | 0.750" | 0.670" | 1.000" | 0.720" | 0.750" | 2.250" | 1.10° | 9.000" |
| CC-SBNEM-1000-8000-SS | 1.000" | 0.890" | 1.500" | 0.970" | 1.000" | 3.000" | 1.50° | 8.000" |

# SCREW-ON MODULAR HEADS

## Dapra's Screw-On Heads Fit Industry Standard Cutting Systems




- Compatible with ISO standard modular cutting systems
- Close-tolerance mounting of heads minimizes runout and maximizes rigidity
- Provide significantly more effective reach than solid end mills
- Use standard inch wrench flats, no special metric wrenches needed

### SBN Screw-On Heads

| Dia. | Holder | M | A | E | Flutes | Wrench |
|---|---|---|---|---|---|---|
| .500" | SBNEM-0500-MOD | M8 | 0.450" | 1.06" | 2 | 3/8" |
| .750" | SBNEM-0750-MOD | M10 | 0.670" | 1.25" | 2 | 9/16" |
| 1.000" | SBNEM-1000-MOD | M12 | 0.890" | 1.63" | 2 | 9/16" |

** M8 modular extensions not available. Use ISO standard bars.*

*See page **22-23** for Modular Extensions.*

# Spare Parts & Tools

| Insert Screw | Thread | Wrenches TORX® | Torque Nm/in.lbs. |
|---|---|---|---|
| SBNS-0500-T20 | M5x.5 | T20-T | 6.0/53 |
| SBNS-0750-T20 | M5x.5 | T20-T | 6.2/55 |
| SBNS-1000-T20 | M6x.75 | T20-T | 6.5/58 |

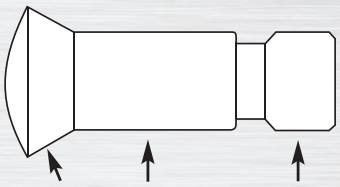
| Miscellaneous | |
|---|---|
| Description | Catalog No. |
| Special Anti-Seize Grease | ASG-120 |

*TORX® is a registered trademark of Camcar/Textron.*

*NOTE: New cutter bodies may require additional torque to fully seat the inserts. Once new cutter pockets are "broken in," the recommended torque specs in the chart can be followed regularly.*


# Correctly Applying Anti-Seize and Indexing Your SBN Insert

1. Anti-seize must be applied before using tool for first time.
2. Remove screw from cutter body.
3. Generously apply anti-seize to *entire length* of screw body, not to just the threads (see diagram).
4. Clean out insert pocket and clean off insert mounting surfaces before assembly of insert/screw combination.
5. Place insert into cutter-body pocket. *If you have already used one edge of your SBN insert, rotate insert 180˚ before placing into pocket.*
6. Place screw with applied anti-seize into position in cutter body.
7. While gently pushing on the end of the TORX® screwdriver/ wrench, begin tightening the screw (may turn with slight resistance in order to pull insert tight into the pocket).
8. Tighten screw to snug fit, taking care not to overtighten. Follow torque specifications shown above.
9. Repeat steps 2-8 for each insert change.
10. Replace screw with each new box of inserts to assure maximum performance.



*Generously apply anti-seize to these surfaces with each insert change.*

# Application Information

## Technical Considerations

- Always use anti-seize compound on threads and screw body.
- Thoroughly clean insert pocket and insert mounting surfaces at each index to maintain best concentricity.
- Change insert screw every 10 inserts (20 edges).
- Use high-quality tool holders for rigidity and concentricity: milling chucks, heat-shrink and mechanical shrink holders are recommended; collets and end mill holders are not recommended.
- Cutter bodies will wear and fatigue over time; inspect tool before each use.

## Recommendations

- Maximum stock removal for semi-finishing should be less than 7% of the SBN diameter.
- Optimum amount of finishing stock to leave is .005-.010"
- Stepover should be greater than or equal to DOC.
- Climb milling is preferred.
- Compensate for Effective Cutting Diameter (see Table 1 and Fig. 1 on p. 19).
- Compensate for chip thinning with Feed Rate Adjustment (see Table 2 on p. 19).
- Surface finish (RMS) is a function of stepover and feed per tooth.
- Try to work within recommended surface footage and chip loads.
- Decrease feed rate coming into corners to reduce chatter.
- For long-reach applications, utilize the Carbide Shank/Carbide Core cutting tools for increased rigidity and reduced chatter.

*DISCLAIMER: Modern metal cutting techniques involve the potential use of very high operating parameters (speeds, feeds, depths of cut, etc.). This creates the potential for flying chips and debris, and can also create tool breakage due to a variety of causes. As such, any metal cutting operation should be executed in a completely enclosed (shielded) environment to protect against injury from flying objects. Dapra does not assume responsibility for any loss, damage or expense incurred in any use or handling of our product after purchase. Grinding produces hazardous dust. To avoid adverse health effects, use adequate ventilation and read material safety data sheet first. This product contains a chemical known to the state of California to cause cancer.*

# Feed, Speed & Diameter Compensation

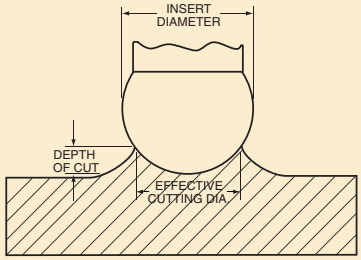
## Table 1: Effective Cutting Diameter (ECD)

### Depth of Cut (DOC)

| Insert Dia. | .005 | .010 | .015 | .025 | .035 | .050 | .100 | .125 | .150 | .200 | .250 |
|---|---|---|---|---|---|---|---|---|---|---|---|
| .500 | .099 | .140 | .171 | .218 | .255 | .300 | .400 | .433 | .458 | .490 | .500 |
| .750 | .122 | .172 | .210 | .269 | .316 | .374 | .510 | .559 | .600 | .663 | .707 |
| 1.000 | .141 | .199 | .243 | .312 | .368 | .436 | .600 | .661 | .714 | .800 | .866 |

1. Select diameter of tool to be used.*
2. Determine Depth of Cut (**DOC**) to be used.*
3. Refer to Figure 1 and Table 1 to find the Effective Cutting Diameter (**ECD**).
4. Refer to Feed and Speed chart on back cover to select the surface footage to be used (**SFM**).
5. Calculate **RPM** using the **ECD** and **SFM** (**SFM** x 3.82 / **ECD** = **RPM**)
6. Refer to Table 2 to determine Feed Rate Adjustment (**FRA**).
7. Refer to chart on back cover to select Feed per Tooth (**FPT**). Calculate Inches per Minute (**IPM**). (**RPM** x **FPT** x 2 x **FRA** = **IPM**)

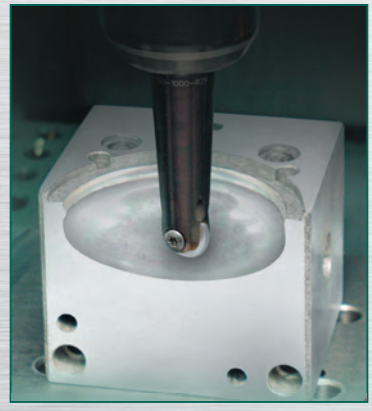
**Figure 1**




** For the cuts that are not strictly tip-cutting, but will use both the tips and sides of the SBN insert, pick an effective cutting diameter in between the ECD in the chart and the actual diameter of the insert. For example: If the ECD comes out to .210" with a .750" diameter insert, use .480" as your ECD when calculating RPM.*

## Table 2: Feed Rate Adjustment (FRA)

*These feed rate multipliers are for applications involving tip-cutting only.*
*For applications including side-cutting or slight/draft angles, lower the multiplier by 20-50%.*



### Insert Diameter

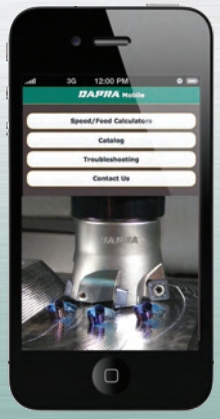
| Depth of Cut (DOC) | 1/2" | 3/4" | 1" |
|---|---|---|---|
| .005 | 5.0 | 6.1 | 7.1 |
| .010 | 3.6 | 4.4 | 5.0 |
| .015 | 2.9 | 3.6 | 4.1 |
| .020 | 2.6 | 3.1 | 3.6 |
| .025 | 2.3 | 2.8 | 3.2 |
| .050 | 1.7 | 2.0 | 2.3 |
| .075 | N/R | 1.7 | 1.9 |
| .100 | N/R | N/R | 1.7 |

*Use multiple above to calculate adjusted feed rate.*

**For optimum speed and feed calculation assistance, visit our mobile website: DAPRA® mobile.com**

**DapraMobile.com** lets you conveniently obtain information about Dapra's extensive line of high-performance cutting tools.

**DapraMobile.com** includes full product specifications, including tool dimensions, insert coatings and grades, recommended cutting speeds and more. You can also access speed & feed calculators to keep critical milling data at your fingertips. Troubleshooting guides are also included to provide solutions to several common milling issues.



# Troubleshooting

| Concern | Possible Cause | Solutions |
|---|---|---|
| **Insert wear at tip** | • Not enough chip load | ❑ Verify correct speed and feed<br>❑ Increase feed rate<br>❑ Decrease RPM<br>❑ Increase DOC |
| **Insert wear appears high (flank wear)** | • Not enough chip load<br>• Surface footage is high<br>• Incorrect grade or coating | ❑ Verify correct speed and feed<br>❑ Increase feed rate<br>❑ Decrease RPM<br>❑ Consider different insert |
| **Insert chipping** | • Surface footage is low<br>• Incorrect grade or coating<br>• Using CB style insert incorrectly<br>• Feed too high | ❑ Verify correct speed and feed<br>❑ Increase spindle speed<br>❑ Decrease feed rate<br>❑ Change insert selection<br>❑ Decrease DOC<br>❑ Use N style insert |
| **Built-up edge on insert** | • Low surface footage<br>• Light chip load (feed per tooth)<br>• Incorrect coating | ❑ Verify correct speed and feed<br>❑ Increase cutting speed<br>❑ Increase feed rate<br>❑ Select different coating |
| **Poor finish/chatter** | • Cutter hung out too far<br>• Excessive runout | ❑ Use Carbide Core cutter body<br>❑ Reduce tool gage length<br>❑ Check tool holder wear |
| **Tool shank breaks** | • Tool pressure too great<br>• Fatigued cutter body | ❑ Decrease DOC<br>❑ Reduce tool gage length<br>❑ Decrease feed rate |

# SBN Cutting Speeds

EDGE²™

## Recommended Cutting Speeds for Dapra SBN Cutters

| MATERIAL GROUP | | Example | UC (uncoated) | GLH (coated) | Recommended FPT |
|---|---|---|---|---|---|
| PLAIN STEELS | < 3%C | 1008, 1018, 12L14 | 300-600 | 800-1600 | .002-.007 |
| | 3%-6%C | 1040, 1045, 1055 | | | |
| | 5%-1.5%C | 1060, 1070, 1095 | | | |
| ALLOY STEELS | Mo | 4012, 4320, 4340 | | 700-1400 | |
| | Cr | 52100, 5120 | | | |
| | NiCrMo | 8620, 8622, 8640 | | | |
| TOOL & DIE STEELS | | A2, D2, P20, W2, H13, S7 | | | |
| HARDENED STEELS | | | N/R | 300-700 | .002-.005 |
| STAINLESS STEELS | Ferritic/ Martensitic | 403, 416, 430, 430F, 434, 446, S44400 | 150-300 | 500-900 | .002-.006 |
| | Austenitic | 304L, 303, 304, 316L | | 300-800 | |
| | Precipitation Hardening (PH) | 15-5PH, 17-4PH, custom, 455, PH13-8 Mo, AM355 | | 250-700 | |
| CAST IRON | Gray | A48 Class xx B, A436 Type 2 | 350-600 | 500-1000 | .003-.007 |
| | Malleable | A47, A220, SAE J148 | | | |
| | Ductile | 60-40-18, 100-70-03, SAE J434 | | 400-900 | |
| ALUMINUM ALLOYS | | 2024-T4, 6061-T6, 7075-T6 | 1000+ | 1000+ | .005-.010 |
| COPPER ALLOYS | CuNi: refer to High-Temp. Alloys below | J463, B121, Ampco 21, Wearite 4-13 | 400-600 | 500-1000 | |
| HIGH-TEMP. ALLOYS | | Inconel 617, Monel K500, Waspaloy, CuNi 70-30 | 50-125 | 75-350 | .002-.004 |
| TITANIUM ALLOYS | | Ti99.9, Alpha Alloy, Ti-6Al-4V | 50-125 | 150-450 | .002-.005 |
| CARBON GRAPHITE | | | 700-1200 | 1200+ | .004-.010 |

*Refer to the Diameter and Feed Rate Adjustment charts on page 19 for accurate RPM and IPM calculations.*

***SPEED***

***Lower Speed Ranges for: Heavier cuts, harder materials, larger-diameter tools***
***Medium Speed Ranges for: Semi-finishing***
***Higher Speed Ranges for: Lighter cuts, softer materials, smaller-diameter tools***

***FEED***

***Lower Feed Ranges for: Heavier cuts, harder materials, smaller-diameter tools***
***Higher Feed Ranges for: Lighter cuts, softer materials, larger-diameter tools***

The parameters provided are suggested operating parameters. Actual speeds and feeds will depend on many variables, such as rigidity, workpiece hardness, tool extension, machine accuracy, Depth of Cut, etc. Start at the middle of the SFM range and the low end of the FPR range. Next, increase FPR to optimize productivity and tool life. Higher SFM will provide higher output but will reduce tool life. Try different combinations to find the parameters that best suit your needs.

## Carbide Core Modular Extensions

### *Dapra's Carbide Core Modular Extensions Are Ideal for Standard Inch End Mill Holders*

- Cylindrical inch shanks, providing adaptation for end mill holders, milling chucks and heat-shrink holders
- 3 sizes to accommodate modular head sizes from $^{3}/_{4}$" to $1^{1}/_{2}$"
- Carbide core for enhanced vibration dampening capability; reduced deflection and improved rigidity
- Optional add-on extensions for additional 2" reach – screw on to base extensions (for $^{3}/_{4}$" to $1^{1}/_{2}$" modular heads)
- Thru-coolant for delivery of air or coolant right at the cutting edge

### Carbide Core Modular Extensions

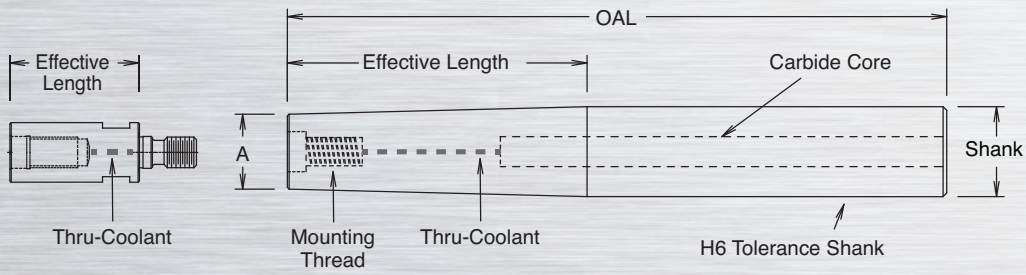
| For Head Dia. | Extension Part No. | Shank Dia. | Effective Length | OAL | Thread | CC | A |
|---|---|---|---|---|---|---|---|
| .750" | CC-ME-0750-3500C-SS | .750" | 3.7" | 5.8" | M10 | 3/8" x 4.0" | 0.660" |
| .750" | CC-ME-0750-3500-C | 1.000" | 3.7" | 6.0" | M10 | 7/16" x 4.0" | 0.660" |
| 1.000" | CC-ME-1000-4500-C | 1.000" | 4.7" | 7.0" | M12 | 7/16" x 5.0" | 0.935" |
| 1.250"/1.500" | CC-ME-1250-5500-C | 1.250" | 5.7" | 8.0" | M16 | 1/2" x 6.0" | 1.175" |

***Extensions feature a cylindrical shank, with no Weldon Flats. Hold with high-performance milling chucks or heat/mechanical shrink holders, or mill Weldon Flats and use a short-length solid end mill holder.***

### 2" Add-On Extensions

| For Head Dia. | Extension Part No. | Effective Length | Thread |
|---|---|---|---|
| .750" | ME-0750-2C EXTENSION ADAPTER | 2.0" | M10 |
| 1.000" | ME-1000-2C EXTENSION ADAPTER | 2.0" | M12 |
| 1.250" | ME-1250-2C EXTENSION ADAPTER | 2.0" | M16 |

***See next page for Heavy Metal Modular Extensions.***




### *Heavy Metal Modular Extensions Provide Even More Cutting Options*

- Made of high-density tungsten, providing extra resistance to vibration and deflection
- Machined on both ends; can be cut in half and used with two different modular heads
- Metric shank diameter provides clearance for each inch size modular head
- Thru-coolant equipped

## Modular Extensions

| Modular Head Dia. | Part No. | OAL | M | Shank Dia. |
|---|---|---|---|---|
| .750" / 20MM | ME-0750-18MM-900-C | 9" | M10 | 18MM |
| 1.000" / 25MM | ME-1000-25MM-1100-C | 11" | M12 | 25MM |
| 1.250" / 1.500" | ME-125/150-25MM-1200-C | 12" | M16 | 25MM |

***Heavy Metal Modular Extensions feature a cylindrical shank, with no Weldon Flats. Hold with high-performance metric milling chucks or heat/mechanical shrink holders, or mill Weldon Flats and use a short-length solid end mill holder.***

# DAPRA® EXPERTISE...

## The difference between a product and a solution

- Indexable Milling
- Direct Part Marking
- Workholding

Dapra's technical expertise and customer support enhance our products' value and performance. Since 1955, we have been in the business of providing innovative applications-based solutions to help you maximize your productivity. Contact us today – our application experts are ready to offer you a complete solution for your productivity requirements.

**DAPRA®**

***Bringing Better Ideas to the Cutting Edge™***

66 Granby Street, Bloomfield, CT 06002

**800-243-3344** • 860-242-8539 • Fax 860-242-3017

Email info@dapra.com • **www.dapra.com**