ALLIED MACHINE & ENGINEERING

Holemaking Solutions for Today's Manufacturing

Technical Guide | Product Overviews and Features

Drilling

Boring

Reaming

Burnishing

Threading

Specials

www.alliedmachine.com

North America

Allied Machine

120 Deeds Drive
Dover, OH 44622
United States

Allied Machine

485 West 3rd Street
Dover, OH 44622
United States

ThreadMills USA™

4185 Crosstowne Ct #B
Evans, GA 30809
United States

Superion®

1285 S Patton St.
Xenia, OH 45385
United States

Europe

Allied Machine Europe

93 Vantage Point
Pensnett Estate
Kingswinford
West Midlands
DY6 7FR, United Kingdom

Wohlhaupter® GmbH

Maybachstrasse 4
Postfach 1264
72636 Frickenhausen
Germany

Asia

Wohlhaupter® India

B-23, 3rd Floor
B Block Community Centre
Janakpuri, New Delhi - 110058
India

Allied Machine & Engineering is a worldwide leader in holemaking and finishing solutions. We are committed to providing practical and dependable solutions to our customers through innovative designs and superior customer and technical support.

We continue to expand our product offering in order to provide new and different solutions. With Field Sales Engineers located around the world, we position ourselves to provide technical support on site, right at your spindle.

ALLIED MACHINE & ENGINEERING®

www.alliedmachine.com

Contents

SECTION

A20

GEN3SYS® XT and XT Pro

WHY SHOULD YOU

GO WITH THE **PRO**?

GEN3SYS® XT Pro

- Increase your penetration rates
- ISO-specific geometries
- Improved chip evacuation
- Increased coolant flow to the cutting zone
- AM420 coating increases heat resistance
- AM440 coating increases abrasion resistance

THAT'S WHY YOU SHOULD

GO WITH THE **PRO**.

Project Profile:	Forged 8640
Tooling Solution:	GEN3SYS XT Pro: P (Steel) Geometry

The Problem:

Previously, the customer was using a competitor drill running at the following parameters:

- 415 SFM (127 M/min)
- 0.009 IPR (0.23 mm/rev)
- The tool drilled a 17.25mm diameter hole to a 20mm depth
- Tool life = **1,000 holes**

The Solution:

Allied Machine recommended the GEN3SYS XT Pro with P (Steel) geometry.

- **Insert** = XTP17-17.25

The tool ran at the following parameters:

- 415 SFM (127 M/min)
- 0.009 IPR (0.23 mm/rev)
- The tool drilled a 17.25mm diameter hole to a 20mm depth
- Tool life = **2,100 holes**

The Advantage:

The GEN3SYS XT Pro increased the tool life from 1,000 holes to 2,100 holes.

Bottom Line: ***Doubled the tool life***

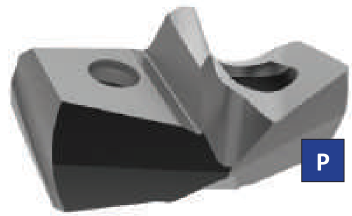
The **PROOF** is in the

NUMBERS

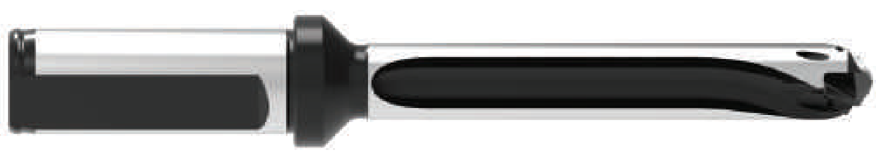
Competitor Insert Tool Life
(number of holes = 1,000)

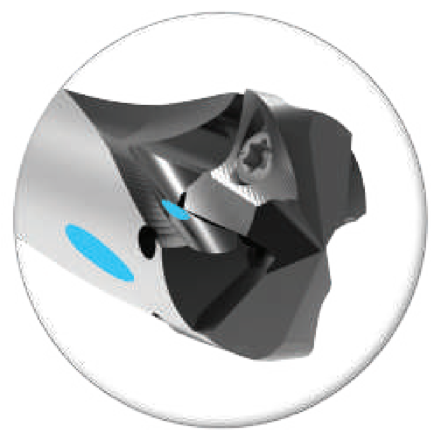
XT Pro Insert Tool Life
(number of holes = 2,100)

2x ***INCREASE*** in tool life



HOLDER DESIGN





Drill deeper holes

The XT Pro holders are available up to 10xD.

- ***This lets you take advantage of the XT Pro insert benefits in deep hole applications.***

Increase your tool life

The coolant configuration increases coolant flow and directs additional coolant to the cutting zone.

- ***This increases tool life with all XT Pro inserts.***

XTP
TAP
TAS
HPU
APX
4TX
REV
OPN
SSD
ACP
BTA
WHL
CRT
ALV
BRN
THM

Competitive Test Results

TEST RESULTS

Project Profile:	Competitive Testing in 4150 Steel
Tooling Solution:	GEN3SYS XT Pro: (P) steel geometry with XT Pro Holder

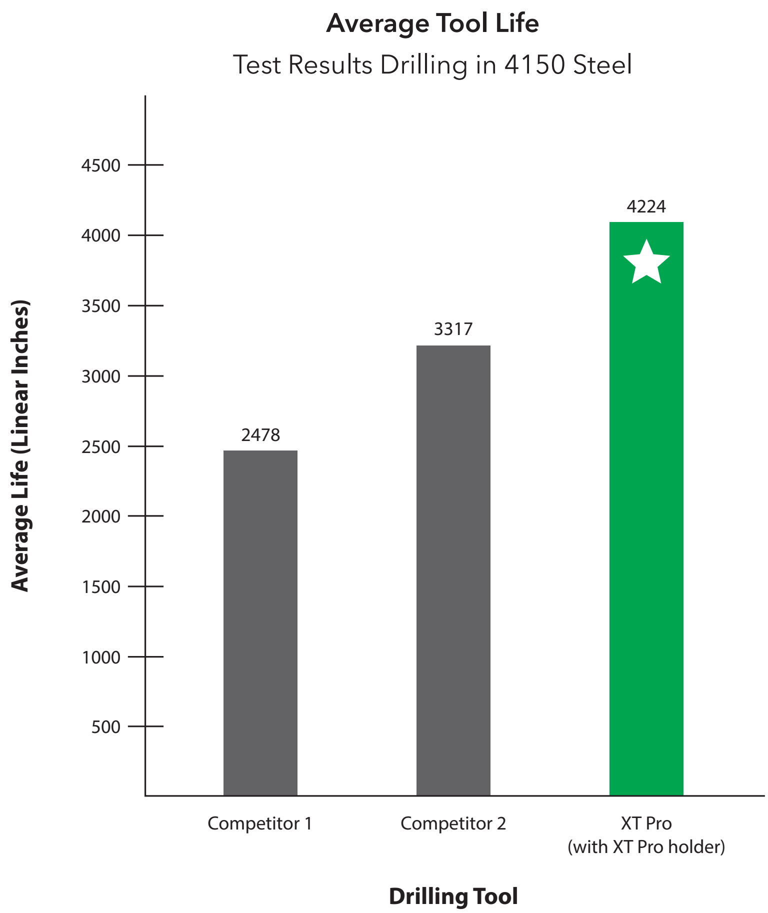
The Parameters:

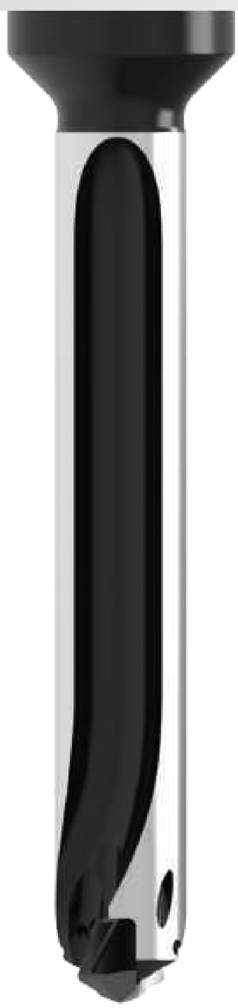
- Hole Diameter = 0.748" (19 mm)
- Depth of Cut = 1-1/2" (38.1 mm)
- Coolant = 300 PSI
- Speed = 1583 RPM
- Feed = 22.16 inch/min (563 mm/min)

The Results:

When run at the listed parameters, here is how the three different tooling solutions performed:

Competitor 1	=	2478 total linear inches
Competitor 2	=	3317 total linear inches
GEN3SYS XT Pro	=	**4224** total linear inches





Case Study Example

CASE STUDY

Project Profile:	Ductile/Nodular Iron
Tooling Solution:	GEN3SYS XT Pro: K (cast iron) geometry

The Problem:

Previously, the customer was using a competitor drill:

- Solid carbide drill
- Tool life = **65 holes**

The Solution:

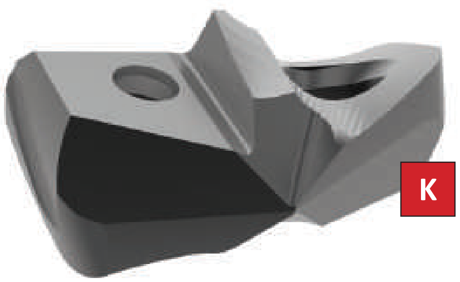
Allied Machine recommended the GEN3SYS XT Pro with K (cast iron) geometry.
The tool ran at the following parameters:

- Hole Diameter = 9/16"
- Coolant = None
- Speed = 390 SFM (117 M/min)
- Feed = 0.008 IPR (0.20 mm/rev)
- Tool life = **390 holes**

The Advantage:

The GEN3SYS XT Pro increased the tool life from 65 holes to 390 holes.

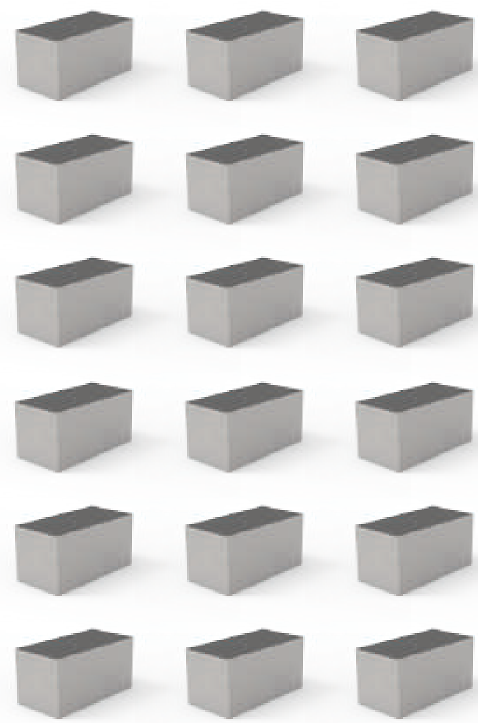
Bottom Line: ***6x the tool life***



The **PROOF** is in the **NUMBERS**

Competitor Tool Life
(number of holes = 65)

XT Pro Tool Life
(number of holes = 390)



There's More to the **Advantage** than Tool Life

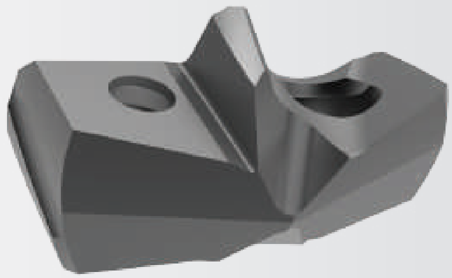
The XT Pro replaceable tip system provides other benefits in addition to the increase in tool life over the solid carbide drill:

- Because only the insert needs changed when it reaches the end of its life, the XT Pro eliminates the need to re-establish tool lengths, which reduces setup times.
- Further benefit in setup is also seen as the tool only needs changed one time for every six of the customer's current method.
- Without the need for regrinds, the customer's stock of tooling is reduced by eliminating the need for float inventory to cover regrind lead time.

6x ***INCREASE*** in tool life

GEN3SYS XT Pro Drilling System Information

GEN3SYS XT Pro Drill Inserts



Advanced Design Capabilities

The advanced XT Pro insert combines a coating and geometry specifically designed to achieve optimal results in ISO material drilling applications. With quick connectivity to existing GEN3SYS drill insert holders, the XT Pro insert can be interchanged with previous XT inserts with ease, resulting in minimal setup times so you can immediately increase your productivity.

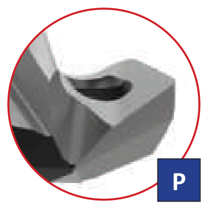
XT Pro Inserts Connect with:

XT Pro holders

GEN3SYS holders

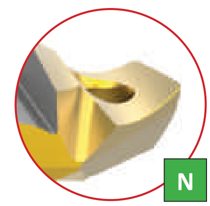
P - Steels

- Designed to provide increased penetration rates and tool life in steel applications
- Superior geometry and edge provides excellent chip control
- Allied's multilayer AM420 coating increases heat resistance and improves tool life



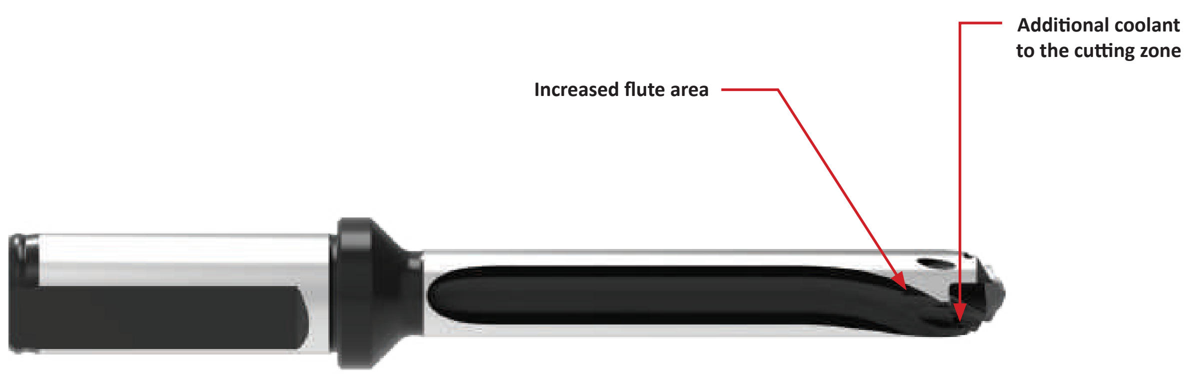
N - Nonferrous Materials

- Designed for applications in aluminum, brass, and copper
- The geometry yields excellent chip control in these softer materials
- TiN coating gives the versatility to run in a variety of materials while reducing buildup

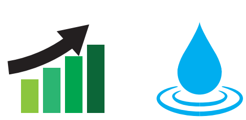
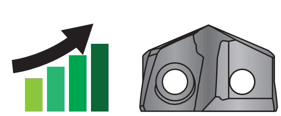


K - Cast Irons

- Uniquely designed for cast/nodular iron applications
- Geometry includes a corner radius for improved hole finish and heat dispersion
- Allied's multilayer AM440 coating provides increased abrasion resistance and tool life

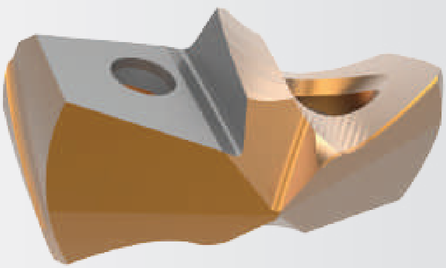


XT Pro Drill Holders

			**3xD, 5xD, 7xD, 10xD**
Straight flutes	Enhanced coolant inlets improve the coolant flow	Provides increased insert life	Available in 3xD, 5xD, 7xD, and 10xD

GEN3SYS XT Drilling System Information

GEN3SYS XT Drill Inserts



High Penetration Drilling Solutions

The unique geometry of the XT inserts provides excellent chip control. They are designed to increase hole quality, surface finish, and true position when compared to other competitive products. The helical margin design provides maximum durability and stability.

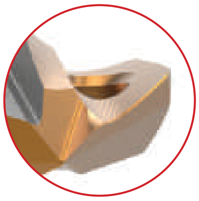
XT Inserts Connect with:

XT Pro holders

GEN3SYS holders

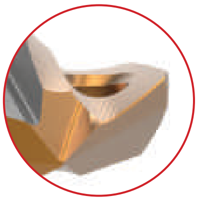
Standard Geometry

- Designed with corner and cutting edge enhancements to deliver more reliability, durability, and productivity
- Increases penetration rates and tool life
- Available in C1 or C2 carbide



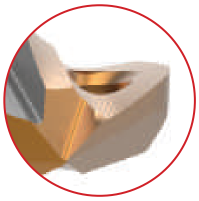
LR - Low Rake Geometry

- The toughest XT geometry available
- Designed for harder steels and less than ideal machining applications
- Available in C1 or C2 carbide



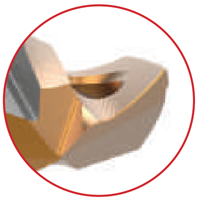
CI - Cast Iron Geometry

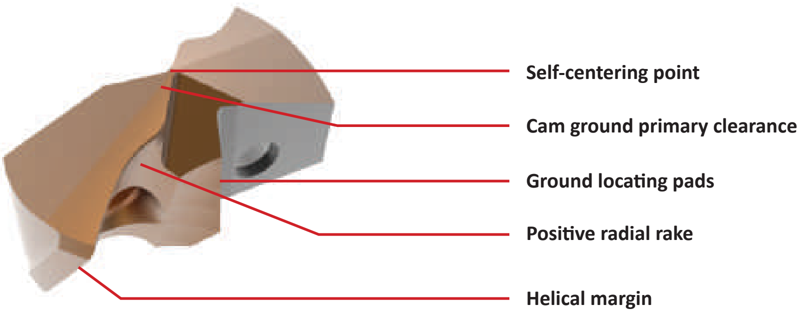
- Increases durability and tool life in ductile, nodular, and grey cast irons
- Available in C2 carbide



AS - Stainless Steel Geometry

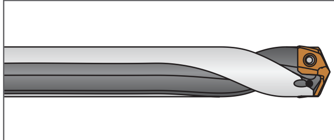
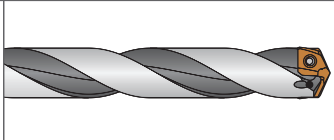
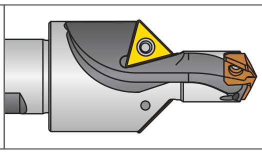
- Designed with a specific geometry to provide unmatched chip control and tool life in austenitic and PH stainless steels, as well as high temperature alloys such as Inconel, Hastelloy, and titanium alloys
- Available in C2 carbide



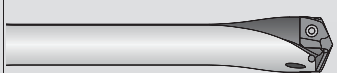


Coating	Features / Benefits
AM300®	• Increased heat resistance over AM200® coating • Up to 20% increased tool life over AM200 coating • Provides superior tool life at high penetration rates

GEN3SYS Holders

			**Stub, 3xD, 5xD, 7xD**
Straight flutes	Helical flutes	Drill / chamfer style	Available in Stub, 3xD, 5xD, and 7xD

Insert Comparison and Assembly Information

		 XT Pro Inserts	 XT Inserts
Recommended for increased productivity		✓	
ISO-specific geometry/coating combination	P N K	✓	
Connects with XT Pro holders		✓	✓
Connects with GEN3SYS holders		✓	✓

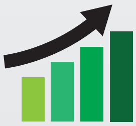
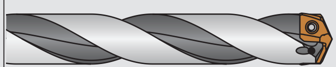
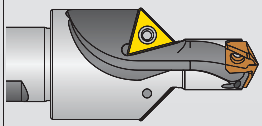
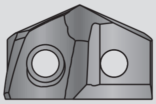
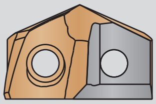
Step 1:
Align the flats on the GEN3SYS XT insert with the flats on the ears of the holder.

Step 2:
Slide the insert into the precision ground locating pocket on the holder. The insert should not be turned, rotated, or twisted for locking purposes. The holder pocket and locating pads on the insert assure optimum fit and repeatability.

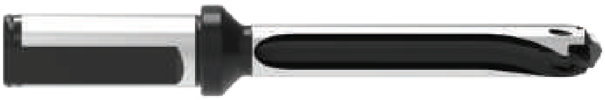
Step 3:
Apply a generous amount of E-Z Break® (provided in the packaging) onto the supplied TORX® Plus screws.

Tighten the TORX Plus screws to the recommended torque value specified in the catalog by series. A preset torx driver is available to assure that the proper torque is applied.

Holder Comparison and Overview

		 XT Pro Holders	 GEN3SYS Holders
Recommended for increased productivity		✓	
Straight flute		✓	✓
Helical flute			✓
Drill/chamfer option			✓
Available in 10xD length	10XD	✓	
Connects with XT Pro inserts		✓	✓
Connects with XT inserts		✓	✓

XT Pro Holders

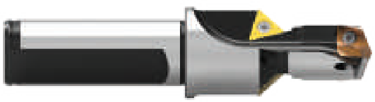


Straight Flute

GEN3SYS Holders

Straight Flute

Helical Flute



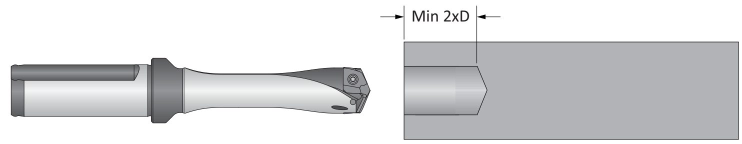
Drill/Chamfer

Deep Hole Drilling Guidelines

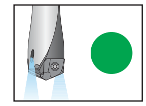
GEN3SYS XT Pro | 10xD Holders

1. Pilot Hole
100 % RPM
100% IPR (mm/rev)

Establish the pilot hole using the same diameter short drill to a depth of 2xD minimum.
Utilize a pilot drill with the same or larger included point angle.

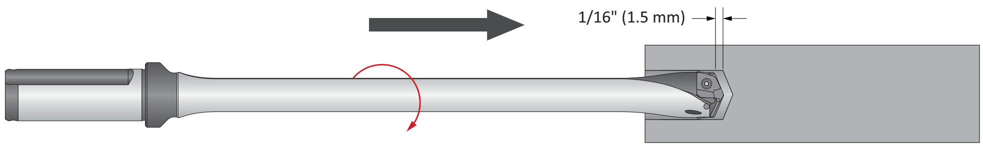


Coolant ON

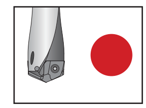


2. Feed-in
50 RPM max
12 IPM (300 mm/min)

Feed the longer drill within 1/16" (1.5 mm) short of the established pilot hole bottom at a **maximum of 50 RPM** and 12 IPM (300 mm/min) feed rate.

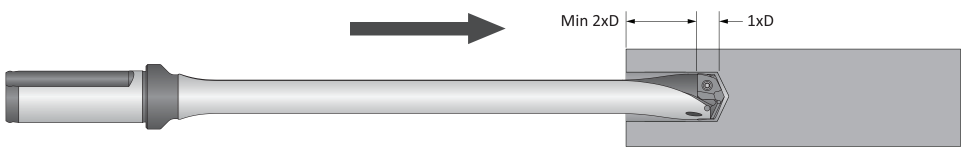


Coolant OFF

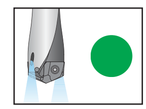


3. Deep Hole Transition Drilling
50 % RPM
75% IPR (mm/rev)

Drill additional 1xD past the bottom of the pilot hole at 50% reduction of recommended speed and 25% reduction of recommended feed.
Minimum of 1 second dwell is required to meet full speed before feeding.

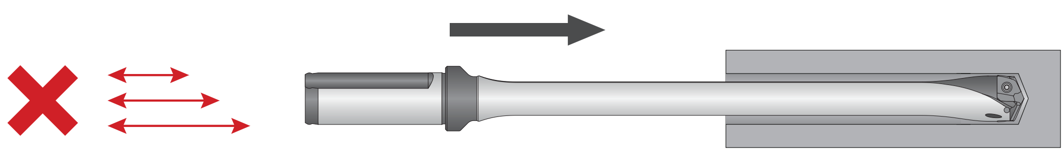


Coolant ON

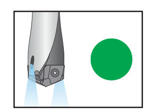


4. Deep Hole Drilling - Blind
100% RPM
100% IPR (mm/rev)

Drill to full depth at recommended speed and feed for longer drill according to Allied speed and feed charts.
No peck cycle recommended.

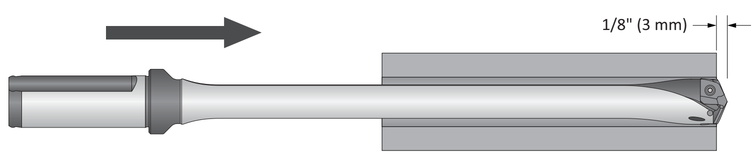


Coolant ON

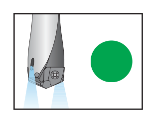


5. Deep Hole Drilling - at Breakout
50% RPM
75% IPR (mm/rev)

For through holes only:
Reduce speed by 50% and feed by 25% prior to breakout.
Do not break out more than 1/8" (3 mm) past the full diameter of the drill.

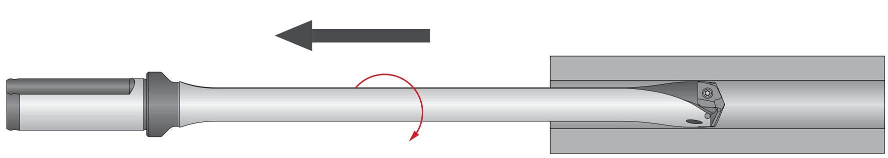


Coolant ON

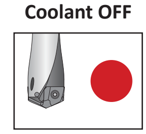


6. Drill Retract
50 RPM max

Reduce speed to a **maximum of 50 RPM** before retracting from the hole.



Coolant OFF



⚠ WARNING **Tool failure can cause serious injury. To prevent:**
- When using holders without support bushing, use a short GEN3SYS holder to establish an initial hole that is a minimum of 2 diameters deep.
- Do not rotate tool holders more than 50 RPM unless it is engaged with the workpiece or fixture.

Visit **www.alliedmachine.com/DeepHoleGuidelines** for the most up-to-date information and procedures. Factory technical assistance is available for your specific applications through our Application Engineering Team. *ext:* **7611** | *email:* **appeng@alliedmachine.com**

Troubleshooting Guide

	Potential Problem																				
	Accelerated corner wear	Barber pole	Bell-mouth hole	Insert chipping	Blue chips	Built-up Edge (BUE)	Chatter	Chip packing	Chipping of point	Damaged or broken tools	Excessive margin wear	High flank wear	Hole lead off	Hole out of position	Hole out of round	Oversize hole	Poor hole finish	Poor tool life	Power spikes - Load meter	Retract spiral	
Setup Condition	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	**Possible Solutions**
Worn or misaligned spindle (lathe, screw machine, chucker)	1		3				7		9	10	11		13			16	17			20	• Align spindle and turret or tailstock. • Repair spindle.
Use of low rigidity machine tools		2	3	4			7		9	10			13	14						20	• Reduce penetration rate to fall within the physical limits of the machine or setup (**NOTICE:** Do not reduce feed below threshold of good chip formation).
Poor work piece support		2		4			7			10	11				15		17			20	• Provide additional support for the work piece. • Reduce penetration rate to fall within the physical limits of the machine or setup (**NOTICE:** Do not reduce feed below threshold of good chip formation).
Flood coolant, low coolant pressure, or low coolant volume	1				5	6		8		10		12				16	17	18	19		• Run coolant through tool holder when drilling greater than 1xD. • Increase coolant pressure and volume through the tool holder. • Reduce penetration rate to fall within the coolant limitations (**NOTICE:** Do not reduce feed below threshold of good chip formation). • Add a peck cycle to help clear chips.
Interrupted cuts. Entry or exit surfaces that are not perpendicular to the spindle (draft angles, parting lines, curved or stepped surfaces, cross holes, and cast or forged surfaces)				4			7		9	10	11		13	14	15	16	17	18			• Premill (spot face) entry or exit surface to remove interruption. • Decrease feed as much as 50% through entry or exit interruption. • Use short holders in low impact entry cuts.
Material harder than expected or running tools beyond recommended speed	1				5	6				10		12						18			• Reduce speed. • Increase coolant pressure and volume. • Improve coolant condition by use of quality products and regular maintenance.
Poor material micro-structure or foreign particles (forgings and castings that have not been normalized or annealed, poorly prepared steel, flame cut parts, and sand casting)				4		6				10		12	13					18			• Compare performance of other tools for similar wear problems, which may indicate poor micro-structure. Anneal or normalize parts to improve micro-structure for machining. • Reduce feeds (**NOTICE:** Do not reduce feed below threshold of good chip formation).
Poor chip control								8		10	11		13			16	17	18	19		• Increase feed to recommended levels. Contact Allied's Application Engineering group for technical recommendations. • Increase coolant pressure and volume. • Improve coolant condition by use of quality products and regular maintenance.
Spot drilled holes with included angle less than that matching GEN3SYS XT or cored holes	1			4			7						13					18			• Spot hole with short tool of same or greater included angle as GEN3SYS XT drill insert. • Reduce feed (**NOTICE:** Do not reduce feed below threshold of good chip formation). If possible, drill from solid.

XTP
TAP
TAS
HPU
APX
4TX
REV
OPN
SSD
ACP
BTA
WHL
CRT
ALV
BRN
THM

Notes

SECTION

A25

T-A Pro® Drilling System

Competitive Test Results

T-A *Pro*®
TEST RESULTS

Project Profile:	Competitive Testing in 4340 Steel
Tooling Solution:	T-A Pro: Steel (P) Geometry with T-A Pro Holder

The Parameters:

- Hole Diameter = 0.5625" (14.30mm)
- Depth of Cut = 2" (50.80mm)
- Coolant = 300 PSI
- Speed = 2546 RPM
- Feed = 16.55 inch/min (420 mm/min)

The Results:

When run at the listed parameters, here is how the 3 different tooling solutions performed:

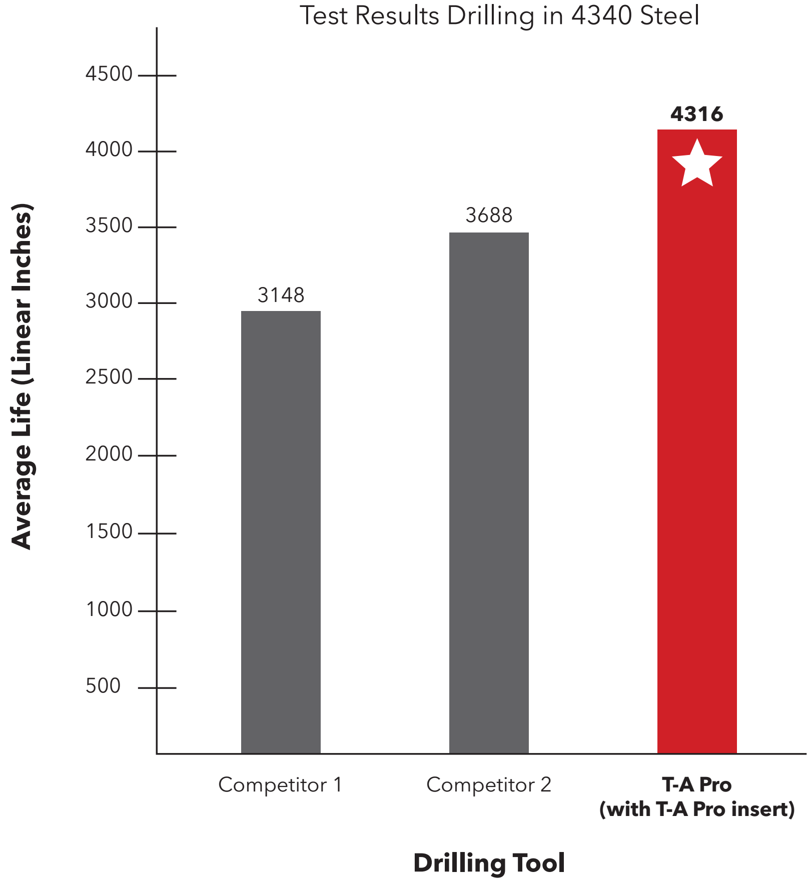
Competitor 1 = 3148 total linear inches

Competitor 2 = 3688 total linear inches

T-A Pro = **4316** total linear inches

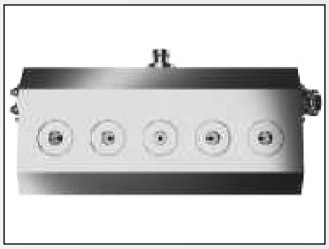
Average Tool Life

Test Results Drilling in 4340 Steel



Case Study

The Gift that Keeps Giving.



Not everything in life has to be a give and take. Our customer who machines fluid end frac blocks was previously having to reduce cutting parameters to achieve good chip formation and produce a successful part.

Needing better chip formation with a reduced cycle time, the customer tested Allied's ***T-A Pro drill***. Using the "M" ISO-specific stainless steel insert geometry–developed for improved chip formation while minimizing exit burr–they were able to increase their speed and feed while maintaining ideal chip formation.

On top of the reduced cycle time, the T-A Pro had a increase tool life lowering the cost per hole by 58.82%. The success of the T-A Pro in this application is just another example of why the T-A Pro is more than just a good drill.

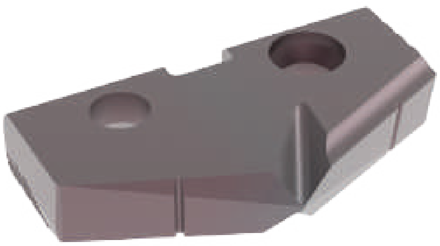
If you are looking for a solution that just keeps giving, ***give us a call, and we will help you find the right solution.***

Product:	T-A Pro drill
Objective:	Reduce cycle time
Industry:	Oil & gas/ Petrochemical
Part:	Fluid end frac block
Material:	15-5 PH Stainless Steel
Hole Ø:	1.75" *(44.45 mm)*
Hole Depth:	20.00" *(508.00 mm)*
Tolerance:	+/- 0.005" *(0.127 mm)*
Required Surface Finish:	125 Ra µin *(3.2 µm)*

Measure	Competitor Drill	T-A Pro Drill
RPM	480	545
Speed Rate	220 SFM *(67.06 M/min)*	250 SFM *(76.20 M/min)*
Feed Rate	0.005 IPR *(0.13 mm/rev)*	0.008 IPR *(0.20 mm/rev)*
Penetration Rate	2.4 IPM *(60.96 mm/min)*	4.4 IPM *(111.76 mm/min)*
Total Part Cycle Time	500 sec	272 sec
Tool Life	30 holes	60 holes
T-A Pro offered 58.82% cost per hole savings over the competitor tooling.		

- T-A Pro holder
  Item No. HTA3D15-150F
- T-A Pro insert
  M geometry (stainless steel)
  Item No. TAM3-44.45

45.60%
cycle time decrease



The ISO-specific AM460 coated T-A Pro insert provided:

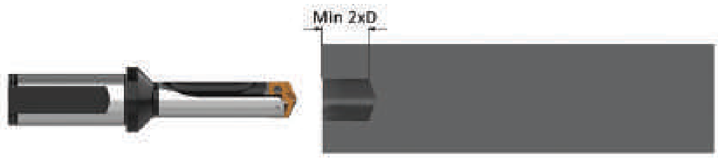
- ✓ Increased tool life
- ✓ Decreased cycle time
- ✓ Decreased cost per hole
- ✓ Increased penetration rate

Deep Hole Drilling Guidelines

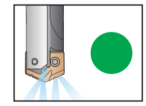
T-A Pro | 10xD, 12xD, and 15xD Holders

1. Pilot Hole
100 % RPM
100% IPR (mm/rev)

Establish the pilot hole using the same diameter short drill to a depth of 2xD minimum.
Utilize a pilot drill with the same or larger included point angle.

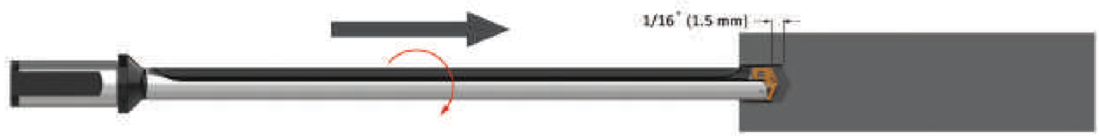


Coolant ON

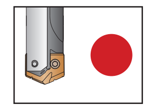


2. Feed-in
50 RPM max
12 IPM (300 mm/min)

Feed the longer drill within 1/16" (1.5 mm) short of the established pilot hole bottom at a **maximum of 50 RPM** and 12 IPM (300 mm/min) feed rate.

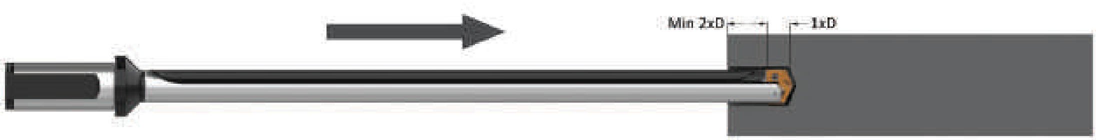


Coolant OFF

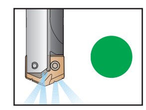


3. Deep Hole Transition Drilling
50 % RPM
75% IPR (mm/rev)

Drill additional 1xD past the bottom of the pilot hole at 50% reduction of recommended speed and 25% reduction of recommended feed.
Minimum of one second dwell is required to meet full speed before feeding.

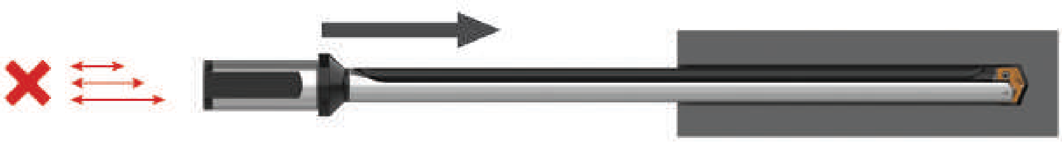


Coolant ON

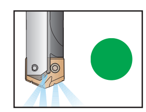


4. Deep Hole Drilling - Blind
100% RPM
100% IPR (mm/rev)

Drill to full depth at recommended speed and feed for longer drill according to Allied speed and feed charts.
No peck cycle recommended.

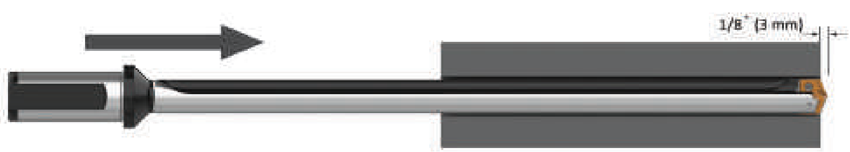


Coolant ON

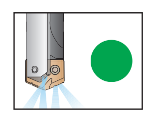


5. Deep Hole Drilling - at Breakout
50% RPM
75% IPR (mm/rev)

For through holes only:
Reduce speed by 50% and feed by 25% prior to breakout.
Do not break out more than 1/8" (3 mm) past the full diameter of the drill.

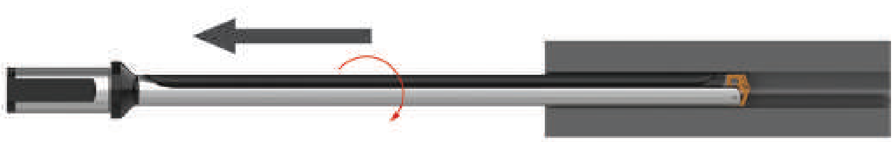


Coolant ON

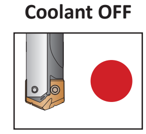


6. Drill Retract
50 RPM max

Reduce speed to a **maximum of 50 RPM** before retracting from the hole.



Coolant OFF



⚠ WARNING **Tool failure can cause serious injury. To prevent:**
- When using holders without support bushing, use a short T-A Pro holder to establish an initial hole that is a minimum of 2 diameters deep.
- Do not rotate tool holders more than 50 RPM unless it is engaged with the workpiece or fixture.

Visit **www.alliedmachine.com/DeepHoleGuidelines** for the most up-to-date information and procedures.
Factory technical assistance is available for your specific applications through our Application Engineering department. *ext:* **7611** | *email:* **appeng@alliedmachine.com**

Troubleshooting Guide

Setup Condition	1 Accelerated corner wear	2 Barber pole	3 Bell-mouth hole	4 Insert chipping	5 Blue chips	6 Built-Up Edge (BUE)	7 Chatter	8 Chip packing	9 Chipping of point	10 Damaged or broken tools	11 Excessive margin wear	12 High flank wear	13 Hole lead off	14 Hole out of position	15 Hole out of round	16 Oversize hole	17 Poor hole finish	18 Poor tool life	19 Power spikes - Load meter	20 Retract spiral	Possible Solutions
Worn or misaligned spindle (lathe, screw machine, chucker)	1		3				7		9	10	11		13			16	17			20	• Align spindle and turret or tailstock. • Repair spindle.
Use of low rigidity machine tools		2	3	4			7		9	10			13	14						20	• Reduce penetration rate to fall within the physical limits of the machine or setup (**NOTICE:** Do not reduce feed below threshold of good chip formation).
Poor work piece support		2		4			7			10	11				15		17			20	• Provide additional support for the work piece. • Reduce penetration rate to fall within the physical limits of the machine or setup (**NOTICE:** Do not reduce feed below threshold of good chip formation).
Flood coolant, low coolant pressure, or low coolant volume	1				5	6		8		10		12				16	17	18	19		• Run coolant through tool holder when drilling greater than 1xD. • Increase coolant pressure and volume through the tool holder. • Reduce penetration rate to fall within the coolant limitations (**NOTICE:** Do not reduce feed below threshold of good chip formation). • Add a peck cycle to help clear chips.
Interrupted cuts. Entry or exit surfaces that are not perpendicular to the spindle (draft angles, parting lines, curved or stepped surfaces, cross holes, and cast or forged surfaces)				4			7		9	10	11		13	14	15	16	17	18			• Pre-mill (spot face) entry or exit surface to remove interruption. • Decrease feed as much as 50% through entry or exit interruption. • Use short holders in low impact entry cuts.
Material harder than expected or running tools beyond recommended speed	1				5	6				10		12						18			• Reduce speed. • Increase coolant pressure and volume. • Improve coolant condition by use of quality products and regular maintenance.
Poor material micro-structure or foreign particles (forgings and castings that have not been normalized or annealed, poorly prepared steel, flame cut parts, and sand casting)				4		6				10		12	13					18			• Compare performance of other tools for similar wear problems, which may indicate poor micro-structure. Anneal or normalize parts to improve micro-structure for machining. • Reduce feeds (**NOTICE:** Do not reduce feed below threshold of good chip formation).
Poor chip control								8		10	11		13			16	17	18	19		• Increase feed to recommended levels. Contact Allied's Application Engineering group for technical recommendations. • Increase coolant pressure and volume. • Improve coolant condition by use of quality products and regular maintenance.
Spot drilled holes with included angle less than that matching T-A Pro or cored holes	1			4			7						13					18			• Spot hole with short tool of same or greater included angle as T-A Pro drill insert. • Reduce feed (**NOTICE:** Do not reduce feed below threshold of good chip formation). If possible, drill from solid.

XTP
TAP
TAS
HPU
APX
4TX
REV
OPN
SSD
ACP
BTA
WHL
CRT
ALV
BRN
THM

Notes

SECTION

A30

T-A® Drilling System

T-A Drilling System Overview | Drill Inserts

Series	Y Series	Z Series	0 Series	1 Series	2 Series	3 Series	4 Series
T-A GEN2							
D_1 inch	0.374 - 0.436	0.437 - 0.510	0.511 - 0.695	0.690 - 0.960	0.961 - 1.380	1.353 - 1.882	1.850 - 2.570
D_1 mm	9.50 - 11.07	11.10 - 12.95	12.98 - 17.65	17.53 - 24.38	24.41 - 35.05	34.36 - 47.80	46.99 - 65.28
Half Series Option*	✖	✖	✔	✔	✔	✖	✖
HSS Substrates	Super Cobalt	Super Cobalt	Super Cobalt	Super Cobalt	Super Cobalt	HSS Super Cobalt Premium Cobalt	HSS Super Cobalt
Carbide Substrates	C1 (K35) C2 (K20)	C1 (K35) C2 (K20)	C1 (K35) C2 (K20)	C1 (K35) C2 (K20)	C1 (K35) C2 (K20)	–	–
Coatings	AM200® AM300®	AM200® AM300®	AM200® AM300®	AM200® AM300®	AM200® AM300®	AM200® TiN	AM200® TiN

*See page A30: 7 for more information regarding half series options

Series	Y Series	Z Series	0 Series	1 Series	2 Series	3 Series	4 Series
T-A							
D_1 inch	0.374 - 0.436	0.437 - 0.510	0.511 - 0.695	0.690 - 0.960	0.961 - 1.380	1.353 - 1.882	1.850 - 2.570
D_1 mm	9.50 - 11.07	11.10 - 12.95	12.98 - 17.65	17.53 - 24.38	24.41 - 35.05	34.36 - 47.80	46.99 - 65.28
Half Series Option*	✖	✖	✔	✔	✔	✖	✖
HSS Substrates	Super Cobalt Premium Cobalt	Super Cobalt Premium Cobalt	Super Cobalt Premium Cobalt	HSS Super Cobalt Premium Cobalt	HSS Super Cobalt Premium Cobalt	Super Cobalt	Super Cobalt
Carbide Substrates	C2 (K20) C3 (K10) C5 (P40) N2	C2 (K20) C3 (K10) C5 (P40) N2	C2 (K20) C3 (K10) C5 (P40) N2	C2 (K20) C3 (K10) C5 (P40) N2	C2 (K20) C3 (K10) C5 (P40) N2	C2 (K20) C5 (P40)	–
Coatings	TiN TiAlN TiCN	TiN TiAlN TiCN	TiN TiAlN TiCN	TiN TiAlN TiCN	TiN TiAlN TiCN	TiN	TiN

*See page A30: 7 for more information regarding half series options

Drill Insert Coatings

AM300®
- Increased heat resistance over AM200® coating
- Up to 20% increased tool life over AM200 coating
- Provides superior tool life at high penetration rates
- Color: copper/orange

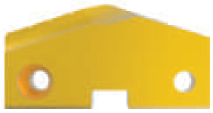
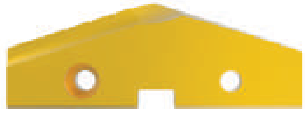
AM200®
- First choice for increased heat resistance over TiN, TiCN, and TiALN with improved wear capabilities
- Allows for improved tool life and higher penetration rates
- Over 20% increase in tool life compared to TiAlN coating
- Color: copper/bronze

TiN
- General purpose coating
- Improved tool life over non-coated inserts
- Excellent choice for aluminum
- Color: gold/yellow

TiAlN
- Excellent choice for wear resistance over high surface speeds
- Excellent oxidation resistance
- Maximum working temperature 800°C
- Color: violet/grey

TiCN
- Excellent choice for wear resistance over low surface speeds
- High hardness/wear resistance
- Maximum working temperature 400°C
- Color: blue/grey

5 Series	6 Series	7 Series	8 Series
			
2.456 - 3.000	3.001 - 3.507	3.508 - 4.000	4.001 - 4.507
62.38 - 76.20	76.22 - 89.08	89.10 - 101.60	101.63 - 114.48
✖	✖	✖	✖
HSS Super Cobalt	HSS Super Cobalt	HSS Super Cobalt	HSS Super Cobalt
–	–	–	–
AM200® TiN	AM200® TiN	AM200® TiN	AM200® TiN

5 Series	6 Series	7 Series	8 Series
			
2.456 - 3.000	3.001 - 3.507	3.508 - 4.000	4.001 - 4.507
62.38 - 76.20	76.22 - 89.08	89.10 - 101.60	101.63 - 114.48
✖	✖	✖	✖
HSS Super Cobalt	HSS Super Cobalt	HSS Super Cobalt	HSS Super Cobalt
–	–	–	–
TiN	TiN	TiN	TiN

Drill Insert Grades			
HSS (T-A / T-A GEN2) First choice for general purpose use. Suited for difficult machining applications with low rigidity, as well as deep hole drilling. Recommended for drilling most steels, cast irons, and aluminum alloys up to 275 BHN.	**HSS Super Cobalt** (T-A / T-A GEN2) Suited for good-to-rigid machining applications, used for drilling exotic and high-alloy materials, or general use when surface speed needs to be increased. For use in material hardness up to 350 BHN.	**HSS Premium Cobalt** (T-A / T-A GEN2) Suited for rigid machining applications, used for drilling exotic and high alloy materials, or general use when surface speed needs to be increased. For material hardness up to 400 BHN.	**Carbide C5 (P40)** (T-A) Excellent for drilling free-machining steel, low/medium-carbon steels, alloy steels, high-strength steels, tool steels, and hardened steels.
Carbide C3 (K10) (T-A) Designed for drilling grey/white cast irons. The special geometry offers substantial increase in penetration rates and provides exceptional edge strength and tool life.	**Carbide C2 (K20)** (T-A / T-A GEN2) Excellent for drilling high-temperature alloys, titanium alloys, cast aluminum, SG/Nodular cast iron, grey/white iron, aluminum bronze, brass, copper, and certain stainless steels.	**Carbide C1 (K35)** (T-A / T-A GEN2) Excellent for drilling free-machining steels, low/medium-carbon steels, alloy steels, high -strength steels, tool steels, and hardened steels.	**Carbide N2** (T-A) Allied's N2 carbide is used with CVD diamond coating. This improves the insert's hardness, durability, and performance, which extends tool life between 30 - 50x over uncoated carbide.

Insert Geometries

There's a Geometry for That

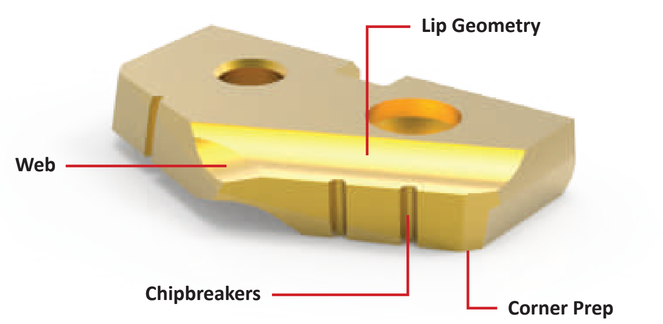
Allied Machine knows there isn't a one-size-fits-all solution when it comes to holemaking. To better accommodate the countless holes our customers drill, we have developed multiple geometry options with new geometries in development at all times.

If you're unsure which geometry would be best for your application, give our Application Engineers a call. They're standing by ready to point you in the right direction.

1.330.343.4283

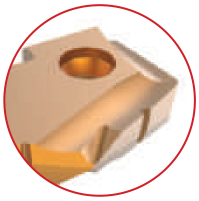
1.800.321.5537 (toll free United States and Canada)

appeng@alliedmachine.com



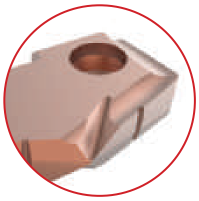
T-A GEN2 Drill Inserts

Standard



- Offers substantial increases in penetration rates and tool life
- Improves centering, drill stability, chip formation, and lowers drill forces
- Provides smoother breakout on through hole applications

High Efficiency (-HE)



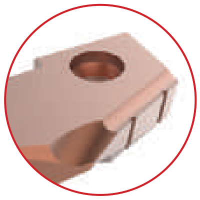
- Excellent chip formation in materials with very high elasticity/ductility and poor chip forming conditions
- Effective in lower-powered machines
- Material example: low carbon steel (not suitable for stainless steel)

T-A Drill Inserts

Standard

- Offers excellent penetration rates and tool life
- Smooth break-out on through holes
- Increases drill stability and chip formation
- Ideally suited for low-to-high rigidity machining applications

Tiny Chip (-TC)



- Unique lip and point design for excellent chip control
- Improved capabilities in long-chipping materials such as low-carbon steels and soft alloy steels
- Enhanced performance in lower-powered machines for better chip formation at lower feed rates

Corner Radius (-CR)

- Improves exit burrs
- Excellent surface finish in most applications
- Improves heat dispersion and tool life
- Can be used in addition to other geometries (as a special)

Special Corner Preparation (-SK)

- Ideal for machining cast iron materials
- Larger than a standard corner clip
- Improves heat resistance
- Standard feature on CI, HI, and HR geometries

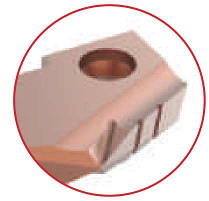
continued on next page

Cam Point (-CP)

- Helical cam ground point
- Improves drill stability and centering characteristics
- Reduces bell-mouthing when using longer holders
- Target materials: steels, cast/forged steels, cast iron

Notch Point® (-NP)

- Reduces bellmouth and lead-off
- Increases stability in deep hole applications
- Reduces thrust
- Can be used in addition to other geometries like cast iron, high rake, and high impact



High Impact (-HI)

- Designed for materials with hardness > 200 BHN (700 N/mm²)
- Enhances chip formation in materials with high elasticity/ductility and poor chip forming characteristics
- SK corner clip improves tool life
- Target materials: structural/cast and forged steels (not suitable for stainless steel)

High Impact Notch Point® (-IN)

- Combination of high impact and Notch Point geometries
- Increases stability in deep hole applications
- Enhances chip formation in materials with high elasticity/ductility and poor chip forming characteristics

High Rake (-HR)

- Designed for materials with hardness < 200 BHN (700 N/mm²)
- Improves chip formation in materials with very high elasticity/ductility, extremely poor chip forming characteristics, and low material hardness
- SK corner clip improves tool life
- Target materials: soft steels, steel castings and forgings (not suitable for stainless steel)

High Rake Notch Point® (-RN)

- Combination of high rake and Notch Point geometries
- Reduces bellmouth and lead-off
- Improves chip formation in materials with very high elasticity/ductility, extremely poor chip forming characteristics, and low material hardness

Cast Iron (-CI)

- Specifically designed for use in grey and white cast irons
- Exceptional edge strength
- SK2 corner preparation for improved tool life
- Standard geometry on C3 (K10) carbide inserts

Cast Iron Notch Point® (-CN)

- Combination of cast iron and Notch Point geometries
- Increases stability in deep hole applications
- Specifically designed for use in grey and white cast irons

Aluminum (-AN)

- First choice for machining aluminum
- Enhanced geometry improves chip formation and hole quality
- TiN coating improves heat resistance and extends tool life

Brass (-BR)

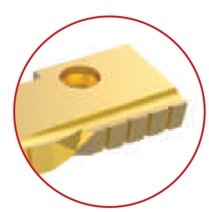
- Improves tool life due to the specialized geometry and edge preparation
- Reduces self-feed tendency

90° Spot and Chamfer (-SP)

- Center cutting web design improves stability and strength
- Eliminates the need for a secondary chamfering operation
- Available with chipbreakers (see -SW below)

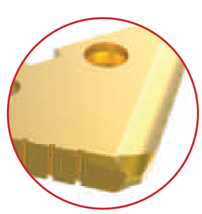
Flat Bottom (-FB)

- Ideal for flattening or squaring the bottom of preexisting holes with high rigidity
- Includes small 10° point on the nose of the insert
- Available without chipbreakers (see -FN below)



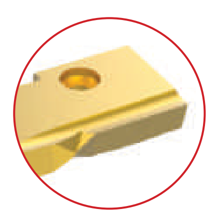
90° Spot and Chamfer (-SW)

- Center cutting web design improves stability and strength
- Eliminates the need for a secondary chamfering operation
- With added chipbreakers



Flat Bottom (-FN)

- Ideal for flattening or squaring the bottom of preexisting holes with high rigidity
- Includes small 10° point on the nose of the insert
- Available with chipbreakers (see -FB above)



Available Standard Insert Geometries

The following table shows which geometries are available as a standard item (based on insert type and series). If you need a geometry on your insert but it is not listed as available, please call the Application Engineering department to discuss quoting your insert as a special to include the desired geometry.

Additional lead time and process fees may apply.

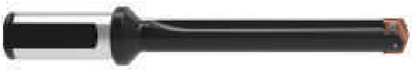
Available Additional Geometries		T-A GEN2			T-A						
					HSS Inserts				Carbide Inserts		
		Y - 2 Series	3 - 4 Series	5 - 8 Series	Y - 2 Series	3 Series	4 Series	5 - 8 Series	Y - Z Series	0 - 2 Series	3 Series
-AN	Aluminum				●				●	●	
-BT	BT-A Specific									●	●
-BR	Brass		●	●	●	●	●	●	●	●	●
-CI	Cast Iron		●		●	●	●		●	●	●
-CN	Notch Point® Cast Iron				●				●	●	●
-CP	Cam Point				●				●	●	
-CR	Corner Radius		●	●	●	●	●	●	●	●	●
-FB	Flat Bottom				●	●	●				
-FN	Flat Bottom				●	●	●				
-HE	High Effciency	●	●								
-HI	High Impact		●	●	●	●	●	●	●	●	●
-HR	High Rake		●	●	●	●	●	●	●	●	●
-IN	High Impact Notch Point®				●				●	●	●
-NC	No Chipbreaker		●	●	●	●	●	●	●	●	●
-NP	Notch Point®				●				●	●	●
-RN	High Rake Notch Point®				●				●	●	●
-SK	Special Corner Preparation		●	●	●	●	●	●	●	●	●
-SP	90° Spot and Chamfer				●	●					
-SW	90° Spot and Chamfer				●	●					
-SS	150° Structural Steel				●	●					
-TC	Tiny Chip				●	●	●	●	●	●	
-TW	Thin Wall				●	●					
-WC	No Corner Clips		●	●	●	●	●	●	●	●	●

Drill Holders

Holder Length Options (for use with both T-A GEN2 and T-A inserts)

Stub Length | Series: Y - 3 (straight flute flanged shank only)

Short Length | Series: ALL



Intermediate Length | Series: ALL

Standard Length | Series: ALL

⚠ **Standard Plus Length** | Series: Y - 2 (helical flute flanged shank only)

⚠ **Extended Length** | Series: 0 - 3

⚠ **Long Length** | Series: 0 - 2

⚠ **Long Plus Length** | Series: 0

⚠ **XL Length** | Series: ALL

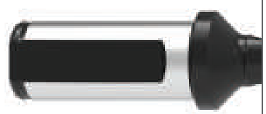
⚠ **3XL Length** | Series: ALL

Holder Shank Options

ER Collet Shank
Series: Y, Z, 0

Straight Shank
Series: ALL

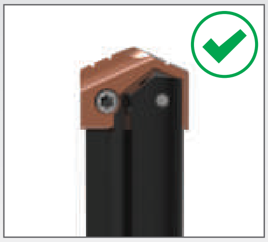
Morse Taper Shank
Series: ALL



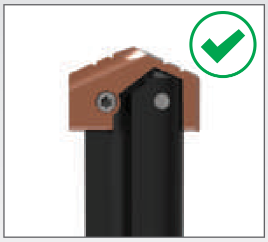
Flanged Shank
Series: ALL

Half Series Holders (0.5, 1.5, 2.5)

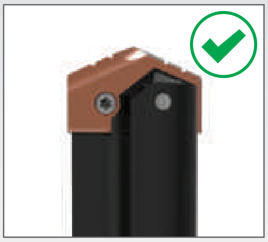
Half series holders are recommended when running carbide inserts toward the upper end of the series drill range, as well as in tougher applications requiring more insert support and holder strength. **NOTE:** Only specified half series inserts should be used with half series holders.



Standard Series Insert +
Standard Series Holder



Half Series Insert +
Standard Series Holder



Half Series Insert +
Half Series Holder

Standard Series Insert +
Half Series Holder

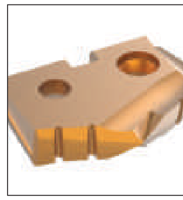
⚠ **WARNING** Refer to Speed and Feed charts for recommended adjustments to speeds and feeds. Refer to page A30: 146 for deep hole drilling guidelines in this section of the catalog. Visit **www.alliedmachine.com/DeepHoleGuidelines** for the most up-to-date information and procedures. Factory technical assistance is available for your specific applications through our Application Engineering Team.

Technical Information

Next Level Solutions: T-A GEN2

What takes a solution to the next level? When you make innovative designs and enhancements to a product that already achieves high performance results, you push the boundaries of what is known. And when you push the known boundaries, the unknown becomes the next level.

After all, everything begins as unknown.

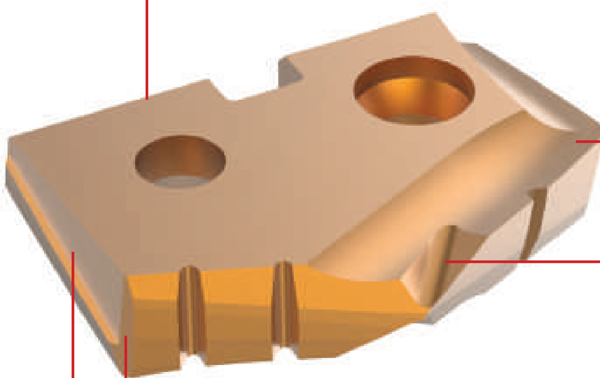


AM300® Coating

- ***Provides superior tool life*** at high penetration rates
- ***Improves heat resistance*** over AM200® coating
- ***Increases tool life*** up to 20% over AM200 coating

AM200® Coating

- ***Improves heat resistance*** over TiN, TiCN, and TiAlN with improved wear capabilities
- ***Increases penetration rates***
- ***Increases tool life*** more than 20% over TiAlN coating



Ground Back Location

- Ensures accurate positioning

Curved Cutting Edge (not all series)

- Enhances chip formation

Notch Point® Geometry

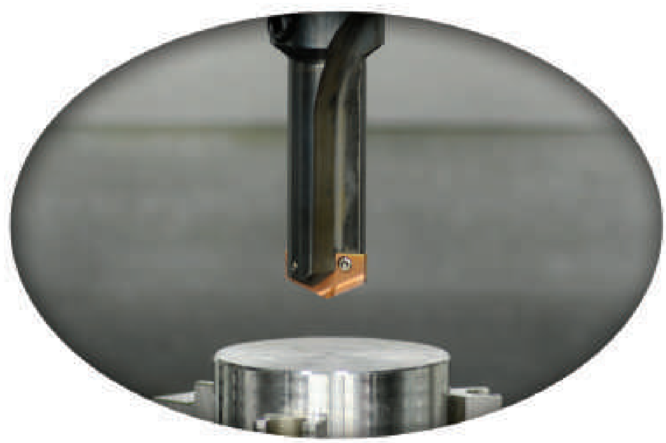
- Improves stability and hole straightness
- Reduces thrust

Corner Clip

- Improves heat dispersion
- Increases tool life

Helical Margin (not all series)

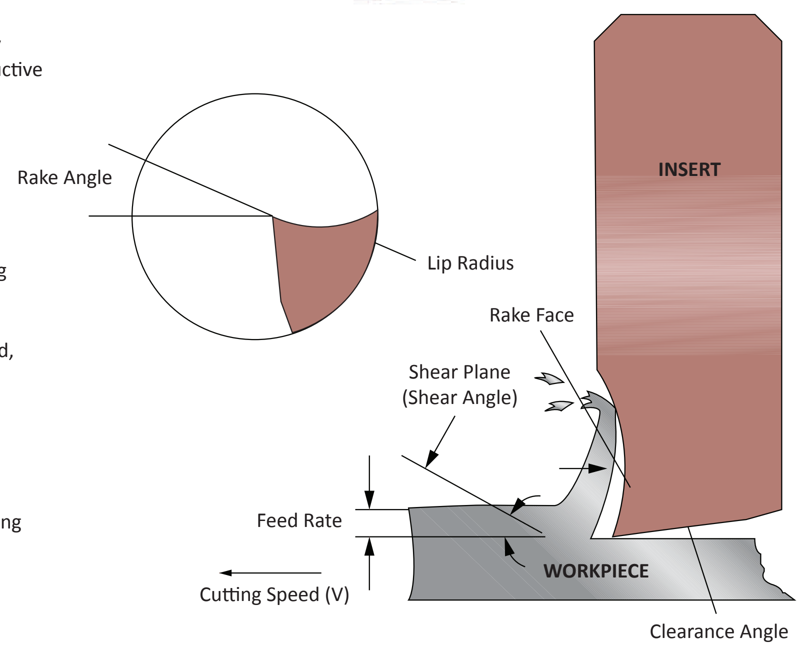
- Increases drill stability



Improving Chip Formation

Achieving optimal chip formation is crucial. The quality of the chips being produced directly affects everything in the entire process: the cycle time, the tool life, the scrap rate, and the quality and condition of the final machined hole.

We know how important chip formation is. That's why we constantly improve and develop new geometries to create a better, more productive T-A product.



Setting Up New Applications

- Check coolant flows adequately through the tool before beginning
- Drill a short hole 1xD deep initially
- The chips produced should be short in length and material colored, not straw or blue
- Measure the hole produced to check that it is within the desired tolerance
- If all is correct, continue to machine the remainder of the hole
- Ensure the drilling process is quiet and smooth with no chip packing

Wear vs Toughness

When selecting a grade of cutting tool material for your application, both wear resistance and grade toughness should be considered. The greater the wear resistance a cutting tool material exhibits, the more likely chipping or fracture is to occur. This requires more rigid machining conditions.

On the other hand, to effectively machine some materials, cobalt or carbide grades of cutting tool material may be required. The graph will aid you in the selection of a cutting tool material with the right combination of wear resistance and toughness to make your application both efficient and cost-effective.

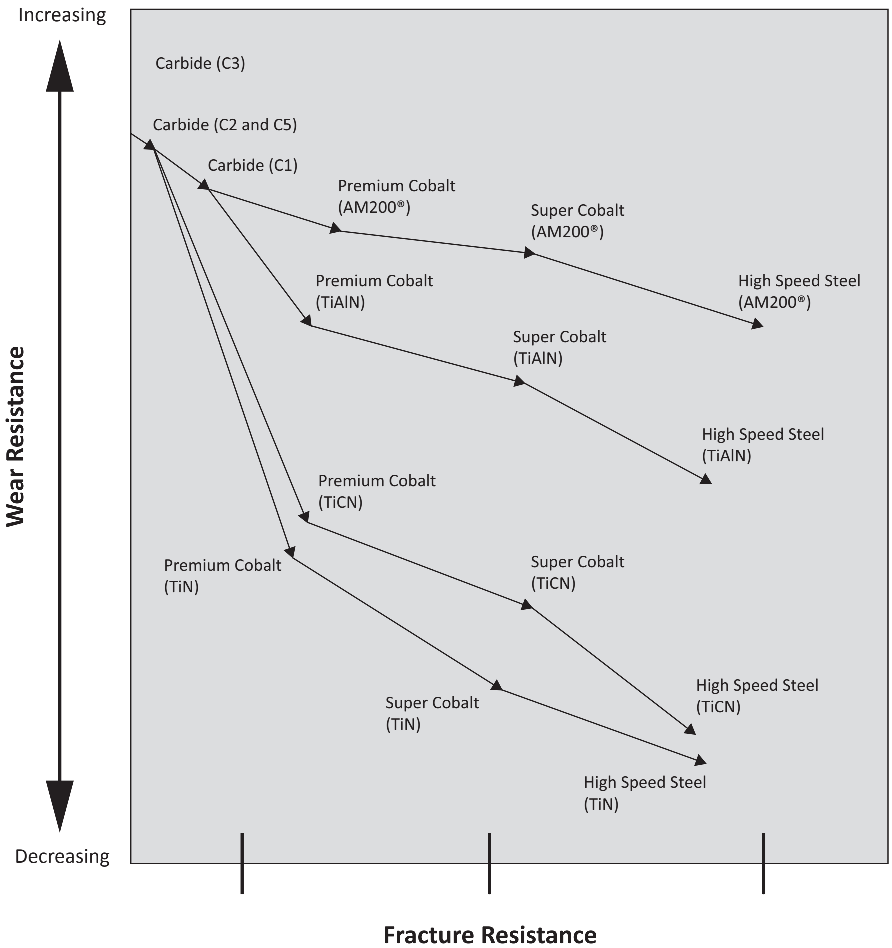

AM300®


AM200®


TiN


TiAlN


TiCN



T-A System Guidelines for Use

- Select the shortest holder possible for the application
- Ensure the T-A holder is held securely and is within 0.003” (0.08mm) of center line
- The T-A insert should be installed in the slot of the holder using the TORX Plus screws provided. These should be tightened to the values listed on the T-A holder pages
- The holder slot should be clean from dirt or debris
- Check that the insert outer diameter is a minimum of 0.012” (0.30mm) larger than the holder body diameter
- Use the recommended cutting data section for guidance when selecting correct insert grades, along with speeds and feeds
- **NOTE:** These cutting parameters are starting conditions only and make no allowance for machine or component rigidity

Troubleshooting Guide

Setup Condition	1 Accelerated corner wear	2 Barber pole	3 Bell-mouth hole	4 Insert chipping	5 Blue chips	6 Built-up Edge (BUE)	7 Chatter	8 Chip packing	9 Chipping of point	10 Damaged or broken tools	11 Excessive margin wear	12 High flank wear	13 Hole lead off	14 Hole out of position	15 Hole out of round	16 Notching of insert	17 Oversize hole	18 Poor hole finish	19 Poor tool life	20 Power spikes - Load meter	21 Retract spiral	22 Step burned on insert	Possible Solutions
⚠ Use of Standard, Standard Plus, Extended, Long, Long Plus, XL, and 3XL holders. See page 146 for Deep Hole Drilling guidelines.		2	3				7		9				13	14			17				21		• Start with short holder and drill a minimum depth equal to 2xD (see page A30: 146 for instructions). • Spot hole with stub tool of same or greater included angle as T-A® drill insert. • Decrease feed a minimum of 50% until establishing full diameter. • Use special holder with wear pads or chrome bearing area to work with drill bushings.
Starting on an inclined surface.							7		9	10	11		13		15						21		• Spot face surface to provide a flat entry surface. • Spot hole with stub tool of same or greater included angle as T-A® drill insert. • Decrease feed a minimum of 50% until establishing full diameter. • Use special holder with wear pads or chrome bearing area to work with drill bushings.
Worn or misaligned spindle (lathe, screw machine, chucker).	1		3				7		9	10	11		13				17	18			21		• Align spindle and turret or tailstock. • Repair spindle. • Spot hole with stub tool of same or greater included angle as T-A® drill insert.
Use of low rigidity machine tools (radial drills, multi-spindle drill press, etc.).		2	3	4			7		9	10			13	14							21		• Spot hole with stub tool of same or greater included angle as T-A® drill insert. • Reduce penetration rate to fall within the physical limits of the machine or setup (**NOTICE:** Do not reduce feed below threshold of good chip formation). • Use special holder with wear pads or chrome bearing area to work with drill bushings. • Use tougher tool steel grades with high wear-resistant coatings.
Poor work piece support.		2		4			7			10	11				15			18			21		• Provide additional support for the work piece. Reduce penetration rate to fall within the physical limits of the machine or setup (**NOTICE:** Do not reduce feed below threshold of good chip formation). • Use tougher tool steel grades with high wear-resistant coatings.
Flood coolant, low coolant pressure or low coolant volume.	1				5	6		8		10		12					17	18	19	20		22	• Run coolant through tool holder when drilling greater than one times diameter. • Increase coolant pressure and volume through the tool holder. • Reduce penetration rate to fall within the coolant limitations (**NOTICE:** Do not reduce feed below threshold of good chip formation). • Add a peck cycle to help clear chips.

⚠ **WARNING** **Tool failure can cause serious injury. To prevent:**

- When using holders without support bushing, use a short T-A® holder to establish an initial hole that is a minimum of 2 diameters deep.
- Do not rotate tool holder more than 50 RPM unless it is engaged with the workpiece or fixture.

Visit **www.alliedmachine.com/DeepHoleGuidelines** for the most up-to-date information and procedures. Factory technical assistance is available for your specific applications through our Application Engineering Team.

	Potential Problem																						
	Accelerated corner wear	Barber pole	Bell-mouth hole	Insert chipping	Blue chips	Built-up Edge (BUE)	Chatter	Chip packing	Chipping of point	Damaged or broken tools	Excessive margin wear	High flank wear	Hole lead off	Hole out of position	Hole out of round	Notching of insert	Oversize hole	Poor hole finish	Poor tool life	Power spikes - Load meter	Retract spiral	Step burned on insert	
Setup Condition	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	**Possible Solutions**
Interrupted cuts. Entry or exit surfaces that are not perpendicular to the spindle (draft angles, stepped surfaces, cross holes, and cast or forged surfaces).				4			7		9	10	11		13	14	15		17	18	19				• Premill (spot face) entry or exit surface to remove interruption. • Spot hole with stub tool of same or greater included angle as T-A® drill insert. • Decrease feed as much as 50% through entry or exit interruption. • Use short holders in low impact entry cuts.
Material harder than expected or running tools beyond recommended speeds.	1				5	6				10		12							19			22	• Reduce speed if a step is worn in the insert, calculate SFM at the worn diameter. Reduce this value by 10% and apply this new value to the original tool diameter. • Increase coolant pressure and volume. • Improve coolant condition by use of quality products and regular maintenance. • Select an insert grade (premium, super cobalt, or carbide) or coating (TiAlN, TiCN, or AM200®) that is more wear-and heat-resistant.
Poor material micro-structure or foreign particles (forgings and castings that have not been normalized or annealed, poorly prepared steel, flame cut parts and sand casting).				4		6				10		12	13			16			19				• Compare performance of other tools for similar wear problems, which may indicate poor micro-structure. Anneal or normalize parts to improve micro-structure for machining. • To improve tool life in materials with poor micro-structure, try carbide grades. • For hard spots or inclusions, use the tougher insert steel grade with high wear-resistant coatings (TiAlN, TiCN, AM200®). • Reduce feeds (**NOTICE:** Do not reduce feed below threshold of good chip formation).
Poor chip control.								8		10	11		13				17	18	19	20			• Increase feed to recommended levels. Contact Allied's Application Engineering team for technical recommendations. • Increase coolant pressure and volume. • Improve coolant condition by use of quality products and regular maintenance. • See pages A30: 4 - 5 for special purpose geometries.
Spot drilled holes with included angle less than that matching T-A® or cored holes.	1			4			7						13			16			19				• Spot hole with short tool of same or greater included angle as T-A® drill insert. • Reduce feed (**NOTICE:** Do not reduce feed below threshold of good chip formation) • If possible, drill from solid.
Use of high wear-resistant insert grades.				4						10													• Use tougher grade of T-A® (from carbide to cobalt to HSS). See wear versus toughness chart on page A30: 9. • Increase rigidity of setup.

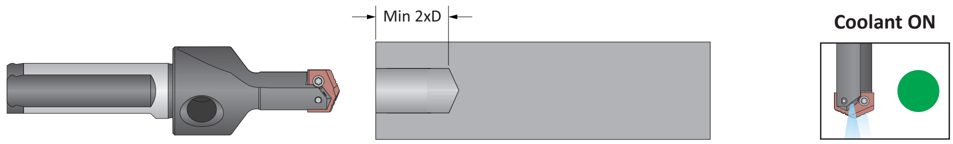
XTP TAP TAS HPU APX 4TX REV OPN SSD ACP BTA WHL CRT ALV BRN THM

Deep Hole Drilling Guidelines

For Lengths Greater Than 9xD (including Standard Plus, Extended Length, Long Length, Long Plus Length, XL, 3XL, and Special Length)

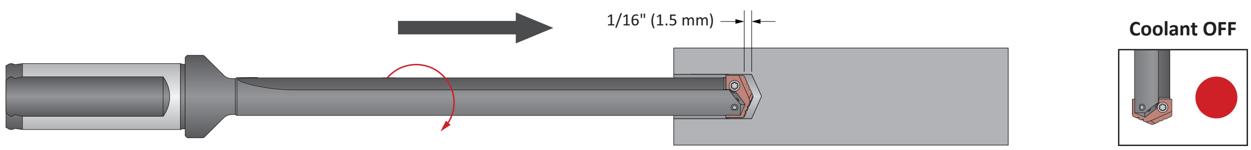
1. Pilot Hole
100 % RPM
100% IPR (mm/rev)

Establish the pilot hole using the same diameter short drill to a depth of 2xD minimum.
Utilize a pilot drill with the same or larger included point angle.



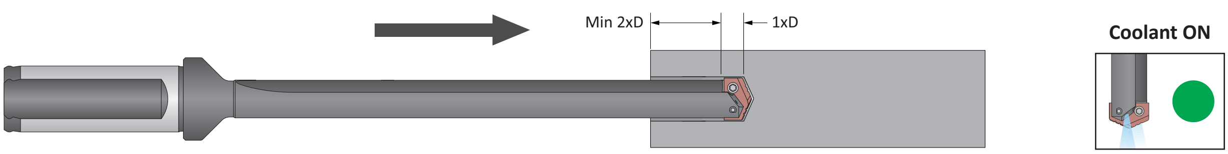
2. Feed-in
50 RPM max
12 IPM (300 mm/min)

Feed the longer drill within 1/16" (1.5 mm) short of the established pilot hole bottom at a **maximum of 50 RPM** and 12 IPM (300 mm/min) feed rate.



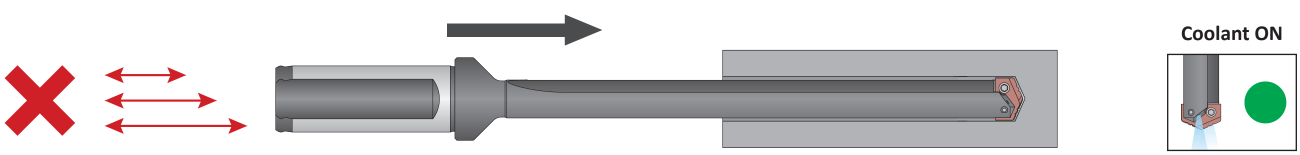
3. Deep Hole Transition Drilling
50 % RPM
75% IPR (mm/rev)

Drill additional 1xD past the bottom of the pilot hole at 50% reduction of recommended speed and 25% reduction of recommended feed.
Mimimum of 1 second dwell is required to meet full speed before feeding.



4. Deep Hole Drilling - Blind
100% RPM
100% IPR (mm/rev)

Drill to full depth at recommended speed and feed for longer drill according to Allied speed and feed charts.
No peck cycle recommended.

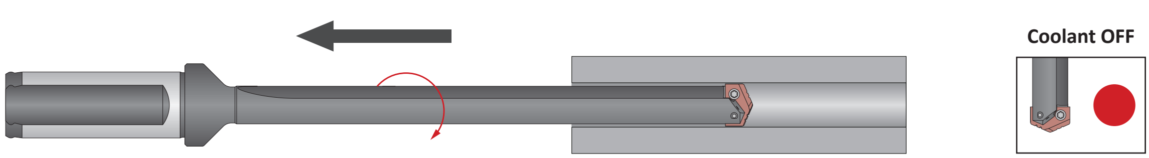


5. Deep Hole Drilling - at Breakout
50% RPM
75% IPR (mm/rev)

For through holes only:
Reduce speed by 50% and feed by 25% prior to breakout.
Do not break out more than 1/8" (3 mm) past the full diameter of the drill.

6. Drill Retract
50 RPM max

Reduce speed to a **maximum of 50 RPM** before retracting from the hole.



⚠ **WARNING** **Tool failure can cause serious injury. To prevent:**
- When using holders without support bushing, use a short T-A® holder to establish an initial hole that is a minimum of 2 diameters deep.
- Do not rotate tool holder more than 50 RPM unless it is engaged with the workpiece or fixture.

Visit **www.alliedmachine.com/DeepHoleGuidelines** for the most up-to-date information and procedures. Factory technical assistance is available for your specific applications through our Application Engineering Team.

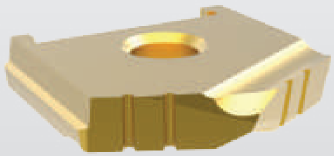
SECTION

A40

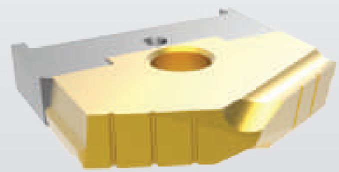
High Performance & Universal

System Overview | Inserts

High Performance Inserts



A - C Series

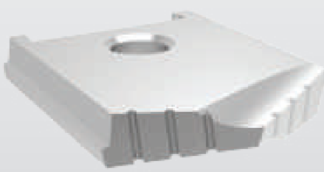


D - H Series
(adapter required)

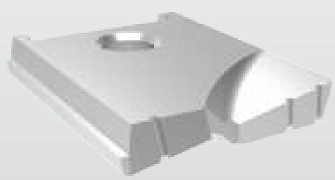
High Performance Inserts

- Increase production 100 - 500% compared to uncoated Universal spade drill inserts
- Fit into Universal style holders
- Available in TiN and TiAlN coatings
- Single-piece design (A - C series) eliminates the need for adapters, which maximizes tool performance in these smaller sizes

Universal Inserts



130° CPM-M4
130° CPM-T15



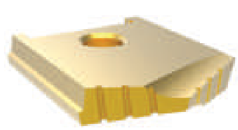
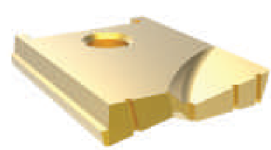
Flat Bottom

90° Spot and Chamfer

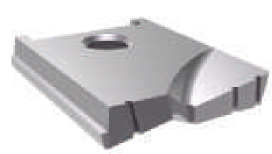
Universal Inserts

- Standard inserts stocked uncoated
- Also available in TiN, TiAlN, and TiCN coatings, which improve tool life when compared to uncoated inserts

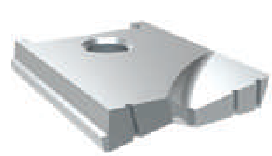
TiN Coating	
Ordering Code: T	Example: 10224-0116**T**


TiAlN Coating	
Ordering Code: A	Example: 10224-0116**A**

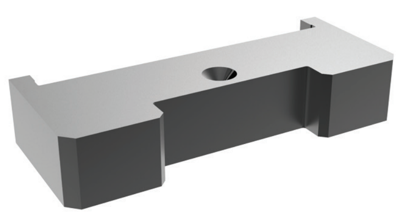

TiCN Coating	
Ordering Code: N	Example: 10224-0116**N**


System Overview | Holders

Straight Shank Holders
- Stub (#125)
- Short (#150)
- Short (#100)
- Standard (#200)
- Long (#250)

Taper Shank Holders
- Short (#300)
- Short (#300 TSC)
- Short (#400 SR)
- Standard (#500 SR)
- Long (#600 SR)
- XL (#700 SR)



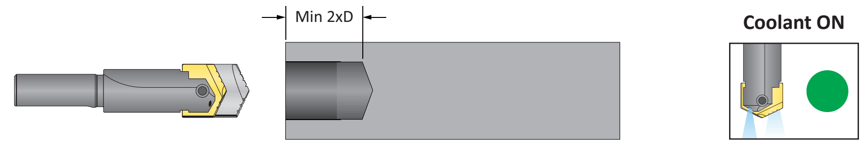
Adapter*
for High Performance D - H series inserts only

*For detailed information and set-up for adapters and Blade-Loc screw assembly, see page A40: 38

Deep Hole Drilling Guidelines

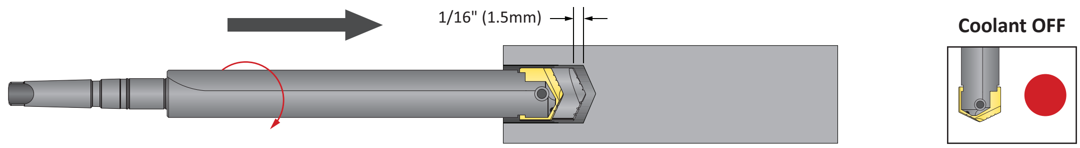
1. Pilot Hole
100% RPM
100% IPR (mm/rev)

Establish the pilot hole using the same diameter short drill to a depth of 2xD minimum.
Utilize a pilot drill with the same or larger included point angle.



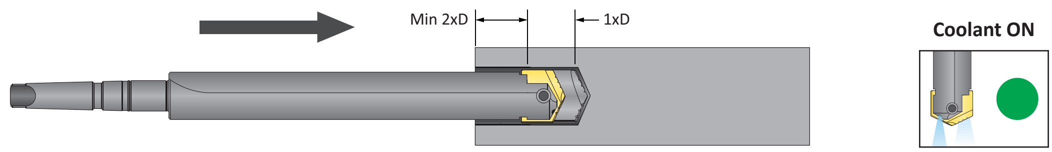
2. Feed-in
50 RPM max
12 IPM (300 mm/min)

Feed the longer drill within 1/16” (1.5mm) short of the established pilot hole bottom at a **maximum of 50 RPM** and 12 IPM (300 mm/min) feed rate.

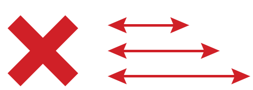


3. Deep Hole Transition Drilling
50% RPM
75% IPR (mm/rev)

Drill additional 1xD past the bottom of the pilot hole at 50% reduction of recommended speed and 25% reduction of recommended feed.
Minimum of 1 second dwell is required to meet full speed before feeding.



4. Deep Hole Drilling - Blind
100% RPM
100% IPR (mm/rev)



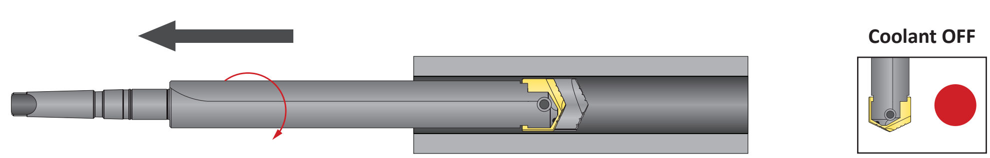
Drill to full depth at recommended speed and feed for longer drill according to Allied speed and feed charts.
No peck cycle recommended.

5. Deep Hole Drilling - at Breakout
50% RPM
75% IPR (mm/rev)

For through holes only:
Reduce speed by 50% and feed by 25% prior to breakout.
Do not break out more than 1/8” (3mm) past the full diameter of the drill.

6. Drill Retract
50 RPM max

Reduce speed to a **maximum of 50 RPM** before retracting from the hole.



⚠ WARNING **Tool failure can cause serious injury. To prevent:**
- When using holders without support bushing, use a short length holder to establish an initial hole that is a minimum of 2 diameters deep.
- Do not rotate tool holders more than 50 RPM unless it is engaged with the workpiece or fixture.

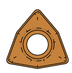
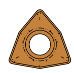
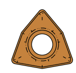
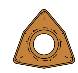
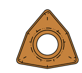
Visit **www.alliedmachine.com/DeepHoleGuidelines** for the most up-to-date information and procedures. Factory technical assistance is available for your specific applications through our Application Engineering Team.

SECTION

A50

APX™ Drill

Drill Selection Guide

Series	33	38	44	51	57
					
Page	6 - 7	8 - 9	10 - 11	12 - 13	14 - 15
D_5 inch	1.299 - 1.496	1.496 - 1.732	1.732 - 2.008	2.008 - 2.244	2.244 - 2.480
D_5 mm	33.00 - 37.99	38.00 - 43.99	44.00 - 50.99	51.00 - 56.99	57.00 - 62.99
ISO Material	P S M H K N	P S M H K N	P S M H K N	P S M H K N	P S M H K N
IC Insert Shape					
IC Insert Size (inch)	5/16"	3/8"	3/8", 1/2"	1/2", 9/16"	9/16"
IC Insert Size (mm)	7.94	9.53	9.53, 12.70	12.70, 14.29	14.29
Wear Pads	NO	NO	NO	NO	NO
Holders					
Drill Depth (inch)	4-7/16 - 14-29/32	5-1/8 - 17-1/4	6 - 20-1/8	6-3/8 - 22-3/8	7-1/8 - 24-3/4
Drill Depth (mm)	112.6 - 378.6	130.5 - 439.9	151.5 - 510.0	161.8 - 570.0	179.9 - 626.9
Pilot Insert					
T-A Series	0, 1	0, 1	1	1	1, 2
GEN3SYS XT Pro Series	–	15, 17, 18, 20	17, 18, 22	18, 20, 22	22, 24, 26

T-A® Style Pilot Insert Head

- Utilizes both T-A Pro and T-A inserts (0 - 2 series)
- Multiple geometry options are available to achieve optimal results in different types of applications

GEN3SYS® XT Style Pilot Insert Head

- Utilizes GEN3SYS XT Pro inserts (15 - 32 series)
- ISO geometry options are available to achieve optimal results in different types of applications

IC Insert AM300®

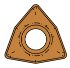
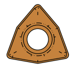
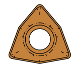
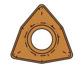
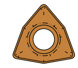
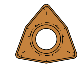
- The design allows for excellent chip control and aggressive penetration rates
- The proprietary AM300 coatings increase tool life above competitors' premium coatings

Insert Application Recommendations

Carbide Grade Options	
C5 (P35)	General purpose carbide grade suitable for most applications. ▶ *Common application in steels and stainless steels.*
C1 (K35)	Toughest carbide grade. Provides the best combination of edge strength and tool life. ▶ *Recommended for less rigid applications.*
C2 (K25)	Higher wear-resistant carbide suitable for abrasive material applications. ▶ *Recommended for grey, ductile, and nodular irons.*
Additional Geometry Option	
High Rake (HR)	Provides superior chip control and tool life in long-chipping carbon and alloy steels below 200 Bhn.

Flanged Straight Shank

CAT40 / CAT50 Integral Shank

63	70	76	83	89	95
					
16 - 17	18 - 19	20 - 21	22 - 23	24 - 25	26 - 27
2.480 - 2.756	2.756 - 2.992	2.992 - 3.268	3.268 - 3.504	3.504 - 3.740	3.740 - 4.000
63.00 - 69.99	70.00 - 75.99	76.00 - 82.99	83.00 - 88.99	89.00 - 94.99	95.00 - 101.60
P S M H K N	P S M H K N	P S M H K N	P S M H K N	P S M H K N	P S M H K N
					
9/16"	3/8"	1/2"	1/2"	9/16"	9/16"
14.29	9.53	12.70	12.70	14.29	14.29
NO	YES	YES	YES	YES	YES
7-7/8 - 27-1/8	8-3/4 - 27-7/8	9-1/2 - 26-1/8	10-1/8 - 27-3/4	10-7/8 - 27-5/8	11-7/8 - 27-1/2
200.8 - 688.3	218.8 - 709.4	239.9 - 664.0	257.8 - 704.9	275.8 - 701.8	302.0 - 698.5
2	2	2	2	2	2
26, 29, 32	29	29	32	29	32

ASSEMBLY

Step 1:

Lower the APX head assembly onto the APX holder.

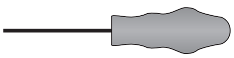
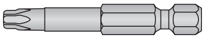
Step 2:

Insert the head mounting screws into points A and B. Tighten until the head is properly secured to the holder.

Step 3:

Tighten with the head mounting driver using the torque setting chart below.

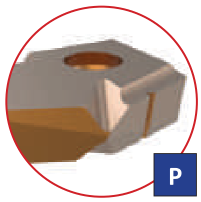
Torque Setting Chart

Series	Screw	Driver	Torque
33 - 63	75020-IP20-1	 8IP-20	60 in-lb (678 N-cm)
70 - 95	78027-IP30-1	 8IP-30B	250 in-lb (2825 N-cm)

Pilot Insert Options

T-A® Pilot Inserts

T-A Pro P - Steels



- Designed to provide increased penetration rates and tool life in steel applications
- Superior geometry and edge provides excellent chip control
- Allied's multilayer AM300 coating increases heat resistance and improves tool life

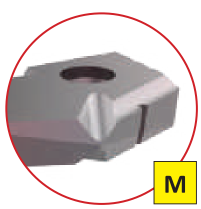
T-A Pro K - Cast Irons

- Uniquely designed for cast/ductile iron applications
- Geometry developed for maximum tool life, reduced exit burr, and improved hole finish
- Allied's multilayer TiAlN coating provides increased abrasion resistance and tool life

T-A Pro N - Non-ferrous Materials

- Designed for applications in aluminum, brass, and copper
- The geometry yields excellent chip control in these softer materials
- TiCN coating gives the versatility to run in a variety of materials while reducing buildup

T-A Pro M - Stainless Steel



- Designed for all stainless steels and heat-resistant super alloys
- Geometry optimized for improved chip formation while minimizing exit burr
- Allied's new AM460 coating provides industry leading tool life in stainless and HRSA materials

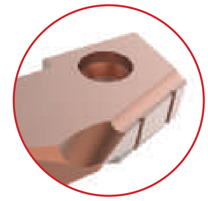
T-A Pro X - High-Speed Steel Materials

- Improved chip geometry for excellent chip control in all materials
- Long tool life and high-process security for the most challenging applications
- Allied's multilayer AM200 coating combines excellent heat resistance and high lubricity for wide application use

T-A Standard

- Excellent choice for general purpose use
- Provides fast penetration rates that produce good hole size and finish
- Combines highly efficient, stable cutting action to minimize power consumption

T-A Tiny Chip (-TC)



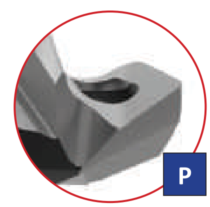
- Unique lip and point design for excellent chip control
- Improved capabilities in long-chipping materials such as low-carbon steels and soft alloy steels
- Enhanced performance in lower-powered machines for better chip formation at lower feed rates

T-A High Impact (-HI)

- Designed to enhance chip formation in materials with high elasticity/ductility and poor chip forming characteristics
- SK2 corner preparation for increased tool life
- Improves chip formation in structural, cast, and forged steels

GEN3SYS® XT Pro Pilot Inserts

P - Steels

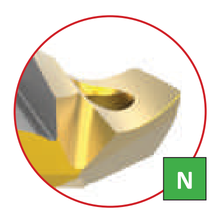


- Designed to provide increased penetration rates and tool life in steel applications
- Superior geometry and edge provides excellent chip control
- Allied's multilayer AM420 coating increases heat resistance and improves tool life

K - Cast Irons

- Uniquely designed for cast/nodular iron applications
- Geometry includes a corner radius for improved hole finish and heat dispersion
- Allied's multilayer AM440 coating provides increased abrasion resistance and tool life

N - Non-ferrous Materials



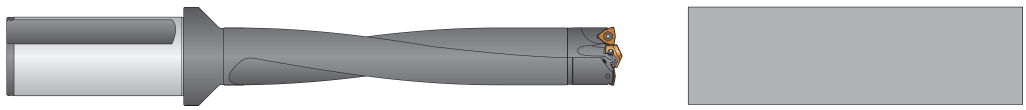
- Designed for applications in aluminum, brass, and copper
- The geometry yields excellent chip control in these softer materials
- TiN coating gives the versatility to run in a variety of materials while reducing buildup

NOTE: For a complete offering of pilot inserts, see sections **A20** (GEN3SYS Drilling Systems), **A25** (T-A Pro Drilling Systems) and **A30** (T-A Drilling Systems) of our catalog.

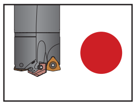
Deep Hole Drilling Guidelines

1. Approach
50 RPM max
12 IPM (300 mm/min)

Feed the longer drill within 1/16" (1.5 mm) short of the workpiece at a **maximum of 50 RPM** and 12 IPM (300 mm/min) feed rate.

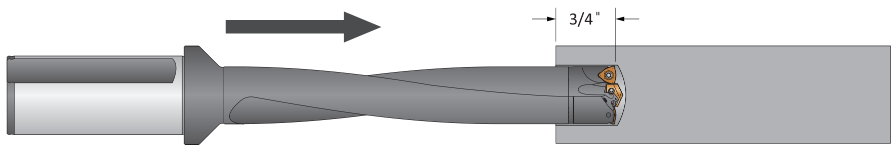


Coolant OFF

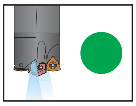


2. Feed-in
Speed at 75% of recommended start
Feed at 50% of recommended start

Drill 3/4" deep at 75% recommended speed and 50% recommended feed to establish the hole.

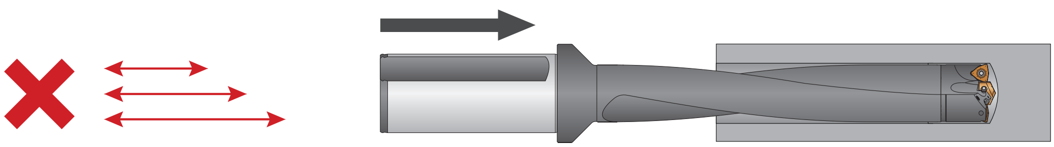


Coolant ON

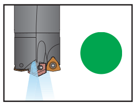


3. Deep Hole Drilling - Blind
100 % RPM
100% IPR (mm/rev)

Drill to full depth at recommended speed and feed for longer drills (according to Allied Machine speed and feed charts).
***No peck cycle recommended.**

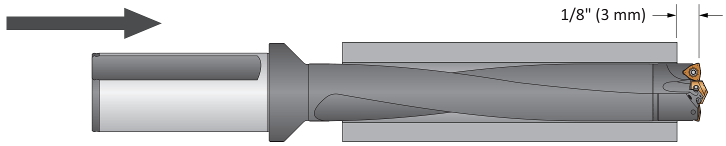


Coolant ON

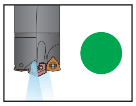


4. Deep Hole Drilling - at Breakout
50% RPM
100% IPR (mm/rev)

***For through holes only:**
Reduce speed by 50% prior to breakout.
Do not break out more than 1/8" (3 mm) past the full diameter of the drill.

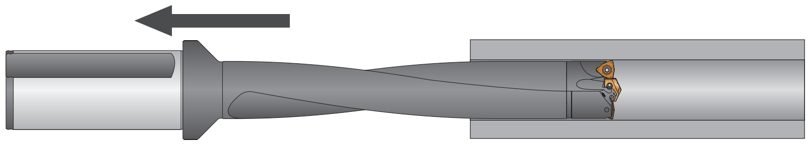


Coolant ON

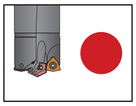


5. Drill Retract
50 RPM max

Reduce speed to a **maximum of 50 RPM** before retracting from the hole.



Coolant OFF



⚠ WARNING **Tool failure can cause serious injury. To prevent:** NEVER rotate these tool holders more than 50 RPM without proper engagement with a workpiece or fixture. Failure to do so could result in tool failure and/or personal injury. Visit **www.alliedmachine.com/DeepHoleGuidelines** for the most up-to-date information and procedures. Factory technical assistance is also available for your specific applications.
ext: **7611** | *email:* **appeng@alliedmachine.com**

XTP
TAP
TAS
HPU
APX
4TX
REV
OPN
SSD
ACP
BTA
WHL
CRT
ALV
BRN
THM

Notes

SECTION

A55

4TEX® Drill

Safety Information

Mechanical / Physical Hazards

Operating cutting tools may present both mechanical and physical hazards. These hazards can result in serious injury to workers or those near machines and damage to machines and the cutting tools. Cutting tools and/or assemblies may break or come loose when in operation causing projectile metal fragments. Metal chips produced by cutting tools have sharp edges and may be very hot. To minimize the risk of mechanical or physical hazards:

- Always secure all components of the cutting tool assembly before operating.
- Wear cut-resistant gloves when handling cutting tool components and assemblies.
- Do not touch metal chips produced by the cutting tools with your hands.
- Always wear appropriate personal protective equipment including safety goggles or glasses with side shields.
- Immediately discontinue use of damaged cutting tools.
- To avoid machine tool damage, make sure the machine has adequate power and torque for the cutting tool when operating. See catalog for power and torque requirements.
- Operating long cutting tools at high spindle speeds can result in a high risk of tool failure and serious injury.
  Visit **www.alliedmachine.com/DeepHoleGuidelines** to read guidelines specific for deep hole drilling.

Dust and Fume Hazards

Grinding, welding, cutting or burning hard metals such as high-speed steel, cobalt or carbides produces hazardous dust and/or fumes. Continued long-term exposure to hazardous dust and fumes can cause serious health issues. To minimize the risk of dust and fume hazards:

- Do not regrind or sharpen cutting tools without using adequate ventilation.
- Use appropriate personal protective equipment such as approved respirator to avoid inhalation, swallowing, or skin contact with the hazardous dust and/or fumes.
- Do not eat, drink, or smoke in the machine operation area. Always wash skin prior to eating, drinking, or smoking to avoid hazardous ingestion.

Sensitizing Hazards

Components of an assembled cutting tool are made from a variety of metal elements that may cause allergic skin reactions with prolonged skin contact. To minimize the risk of allergic skin reactions:

- Avoid skin contact with cutting tools.
- Wear appropriate gloves and protective clothing.
- Wash skin and launder clothing after handling cutting tools to reduce the risk of skin allergies.

Preventive Safety Measure Applicable to all Hazards

- Prior to using cutting tools, always read Allied Machine's Safety Data Sheets, product catalog, and product labels for additional warnings for the Allied Machine product being used.
- For machining safety, only operate equipment when all necessary guards, interlocks and other safety devices are in place and functional. Use all appropriate safety guards or machine encapsulations to securely collect particles such as chips or cutting elements that may become projectiles.

Through Hole

- With through holes, a **sharp-edged disk** is created as tool breakout occurs.

  ⚠ Proper personal protective equipment must be used to prevent injury (e.g. wear cut-resistant gloves).

Case Study

Do you need performance in extreme machining conditions?

Tooling is only a sliver of the pie when it comes to productivity. It doesn't matter what your tooling is capable of if your machine conditions restrict those capabilities. Our customer, who drills holes for machine gun bolt switches, utilizes a machine with oil coolant that creates more extreme drilling conditions than water-based coolant.

Because oil coolant doesn't dissipate heat fast enough, the customer's tooling only lasted for 160 holes per insert, and the tool experienced sporadic failure. They also needed to run a peck cycle for chip control.

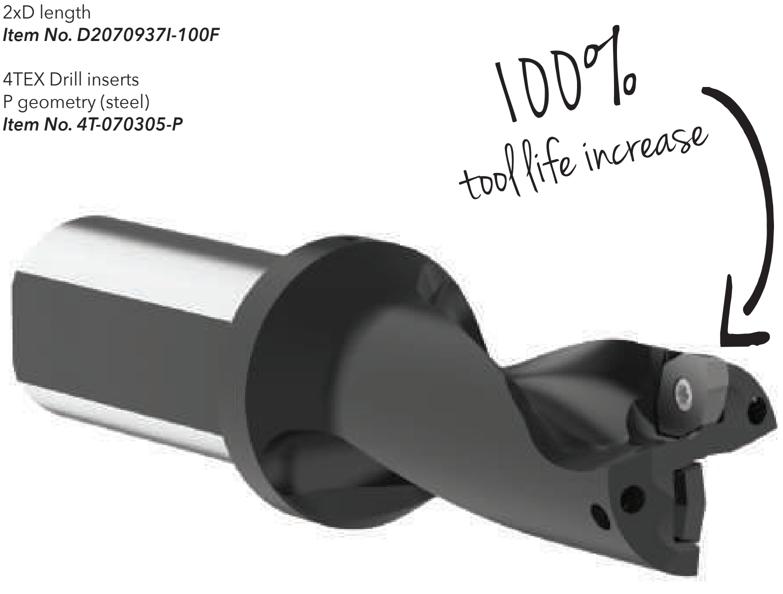
The customer decided to test the **4TEX indexable carbide drill** using the "P" geometry with AM480 coating designed specifically for wear-resistance in steel material applications. The 4TEX "P" geometry allowed for the speed and feed to be altered and accommodated the machine's oil coolant. The 4TEX penetration rate was able to decrease cycle time and also double the tool life to 320 holes per insert. The 4TEX geometry also improved chip formation and eliminated the peck cycle.

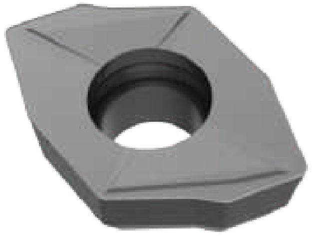
The 4TEX provided the stable and repeatable process the customer was looking for while increasing tool life by 100%. With all their objectives met, the customer was thrilled with the solution that optimized their machine's limitations. ***Are you using the solution that best optimizes your machine's limitations?***

Product:	4TEX® Drill
Objectives:	(1) Exceed 160 holes per insert (2) Eliminate peck cycle (3) Provide stable/repeatable process
Industry:	Firearms
Part:	Machine gun bolt switch hole
Material:	4340 steel
Hole Ø:	0.937" *(23.7998 mm)*
Hole Depth:	0.590" *(14.986 mm)*

Measure	Competitor IC Drill	4TEX® Drill
RPM	2075	1223
Speed Rate	509 SFM *(155.1432 M/min)*	300 SFM *(91.44 M/min)*
Feed Rate	0.0015 IPR *(0.0381 mm/rev)*	0.003 IPR *(0.0762 mm/rev)*
Penetration Rate	3.11 IPM *(78.994 mm/min)*	3.67 IPM *(93.219 mm/min)*
Peck Cycle	Yes	No
Cycle Time	16 sec	9 sec
Tool Life	160 holes per insert	320 holes per insert

- 4TEX Drill holder
  2xD length
  Item No. D2070937I-100F
- 4TEX Drill inserts
  P geometry (steel)
  Item No. 4T-070305-P





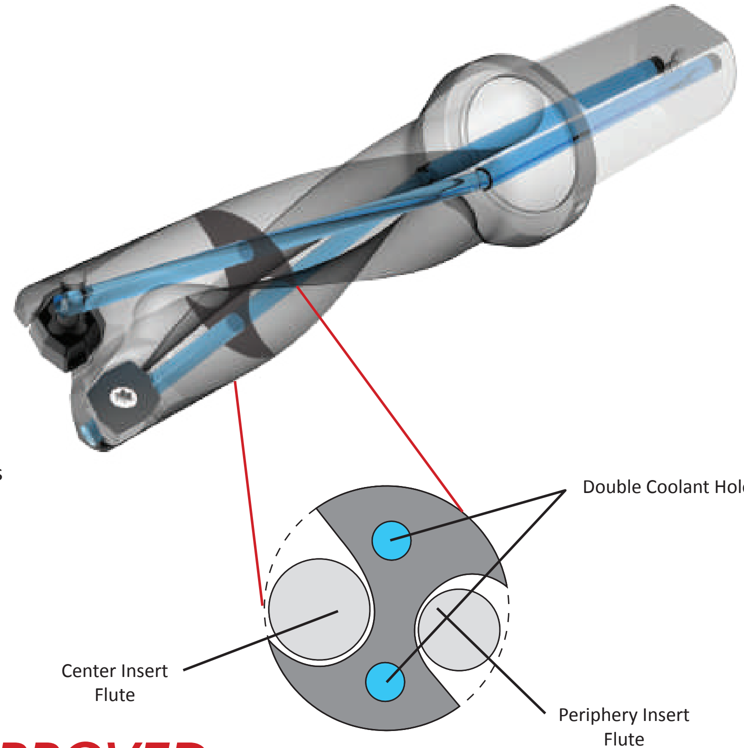
The 4-sided indexable inserts with wear-resistant coating provided:

- ✓ Increased tool life
- ✓ Decreased cycle time
- ✓ Worry-free machining

Product Overview

4TEX Drill ***Advantages***

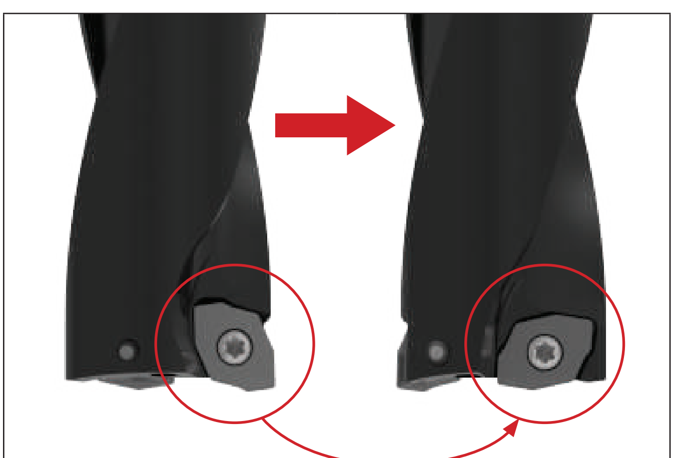
- ***Superior chip evacuation***
  provided by the two twisted coolant holes
- ***Improved hole size***
  from the increased holder rigidity
- ***Longer tool life***
  provided by the four-sided insert design
- ***Optimal chip formation***
  with ISO-specific insert geometry/coating combinations
- ***Competitive cycle times***
  due to single effective cutting when using light duty machines



Designed TO GIVE YOU IMPROVED HOLE SIZE AND STRAIGHTNESS

- The two twisted coolant holes allow the core to remain intact, making the core thicker and stronger for improved hole straightness even in uneven surfaces.
- The enlarged dual coolant outlets increase the coolant volume, which improved the chip evacuation resulting in improved hole size.
- The flute space of the internal cutting edge side (where chips get stuck most often) is 1.6x larger than typical IC drills, helping to mitigate catastrophic failures and improve hole size.

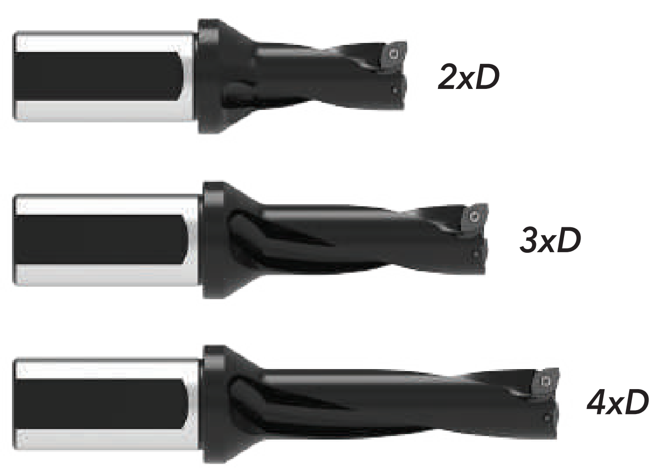
LONGER TOOL LIFE



The 4 Edge Advantage

When the cutting edge on the periphery insert wears down, it can be rotated and moved to the center insert position.

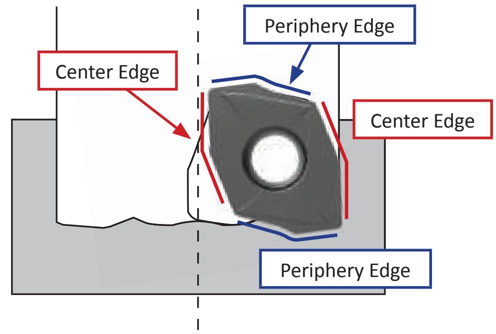
AVAILABLE *LENGTHS*



Insert Information

4 CUTTING EDGES

- Each insert has two inner cutting edges and two outer cutting edges
- Economical solution that increases tool life because of the rotation ability of the inserts
- Available in ISO material-specific geometry/coating combinations

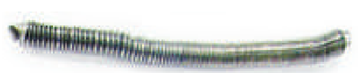


Periphery Insert

Center Insert

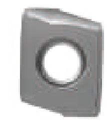
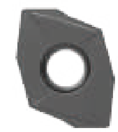
Periphery edge chip formation:

Center edge chip formation:

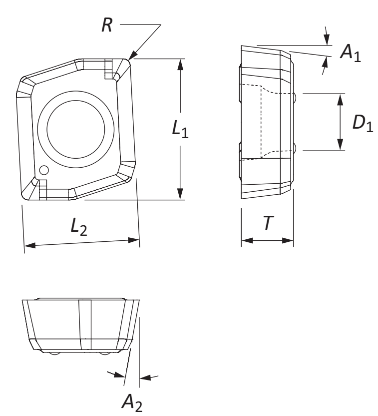


ISO Material	Geometry	Coating	Description
P	General Rake	AM480	A general purpose geometry that provides excellent chip formation in most steels including free-machining, medium- and high-carbon steels. A P30 carbide substrate for improved toughness and AM480 coating, a proprietary wear resistant multilayer PVD coating to improve tool life.
S M	High Rake	AM485	A higher rake geometry that provides excellent chip formation in both stainless steels and high-temperature alloys. A tough M25 carbide substrate coated with AM485, a high heat resistance proprietary multilayer PVD coating.
H	Low Rake	AM480	A lower rake geometry to improve edge strength in both hardened tool steels and high-strength alloys. With a P30 carbide substrate for improved toughness and coated with AM480, a proprietary multilayer PVD coating to improve resistance against tool wear.
K	General Rake	AM480	With a general purpose geometry, the K inserts can be used in grey cast irons as well as ductile irons. A high wear-resistant K10 carbide substrate to improve tool life and coated with AM480, a proprietary multilayer PVD coating to improve resistance against tool wear.
N	High Rake	TiCN	A higher rake cutting geometry provides excellent chip formation in non-ferrous materials. An M15/K15 carbide substrate paired with TiCN coating for improved lubricity to resist built-up material, increasing tool life and maintaining chip formation.

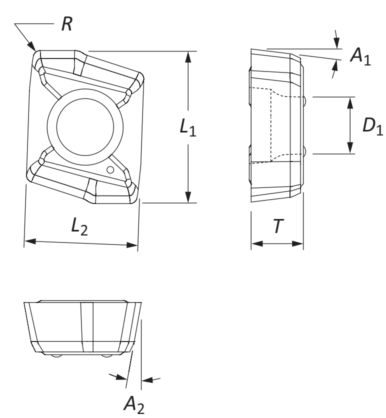
Insert Information

Series	Insert Prefix	Dimension (mm) L_1	L_2	T	D_1	R	Angle A_1	A_2	Shape
03	**4T-030203C-x**	5.60	4.80	2.30	2.40	0.30	7°	10°	 Style 1
	4T-030203P-x	6.38	4.77	2.30	2.40	0.30	7°	10°	 Style 2
04	**4T-040203-x**	6.21	5.06	2.60	2.45	0.30	13°	10°	 Style 3
05	**4T-05T203-x**	7.26	5.48	2.76	2.55	0.30	13°	7°	
06	**4T-06T204-x**	8.59	6.44	2.89	2.79	0.40	13°	7°	
07	**4T-070305-x**	10.21	8.02	3.24	3.00	0.50	13°	7°	
09	**4T-09T306-x**	12.18	9.55	4.03	3.64	0.60	13°	7°	
11	**4T-11T306-x**	14.50	11.61	4.06	4.62	0.60	13°	7°	
14	**4T-140408-x**	17.99	14.40	4.88	5.76	0.80	13°	7°	

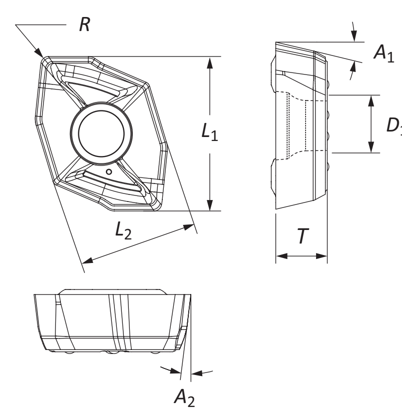
Style 1



Style 2



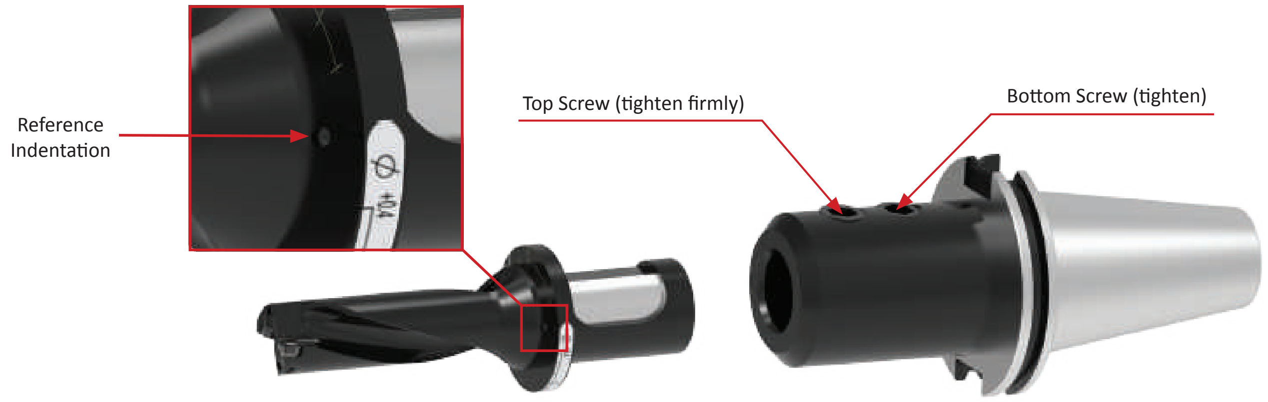
Style 3



XTP
TAP
TAS
HPU
APX
4TX
REV
OPN
SSD
ACP
BTA
WHL
CRT
ALV
BRN
THM

Diameter Adjustment

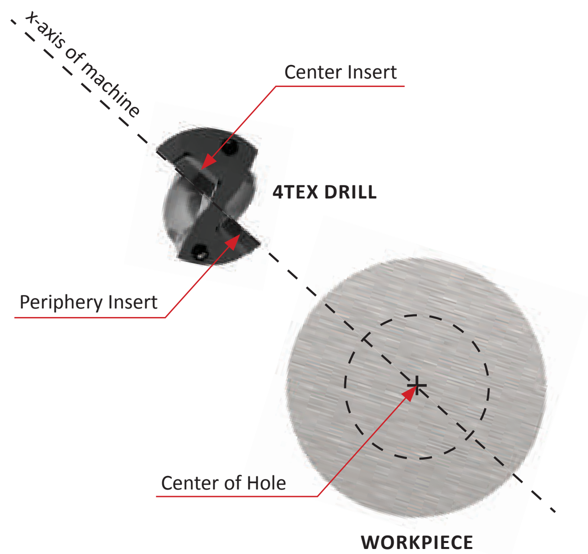
Milling and Lathe Applications

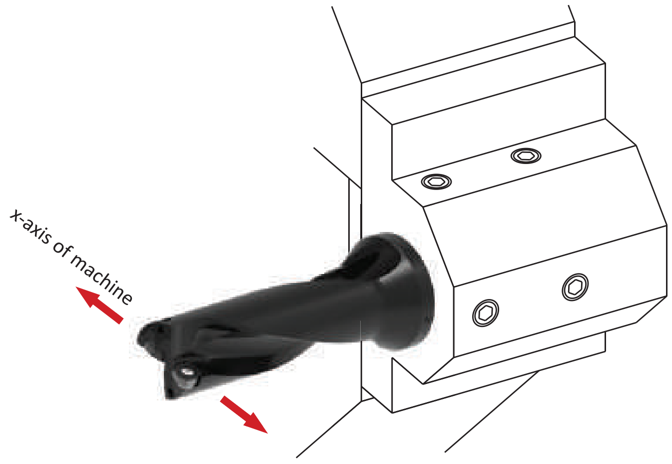


For Milling Applications

1. Assemble the 4TEX drill, eccentric sleeve, and tool holder. Do not tighten the tool holder set screws.
2. Using the peripheral marks for milling machines, align the reference indentation on the holder with the 0 (zero) mark on the eccentric sleeve to have no offset.
3. Rotate the sleeve in the (+) or (-) direction to increase or decrease the nominal diameter.
4. Once the drill has arrived at the desired diameter, firmly tighten the top set screw first and then tighten the bottom set screw.

NOTICE: Eccentric sleeves are to be used with side-locking tool holders only. Damage may result with other styles of tool holders.





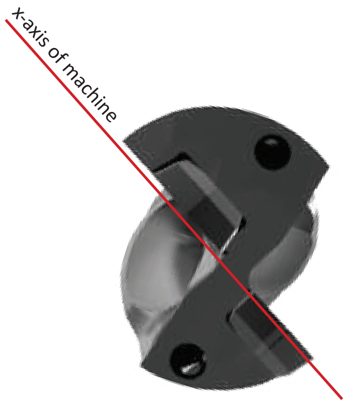
For Lathe Applications

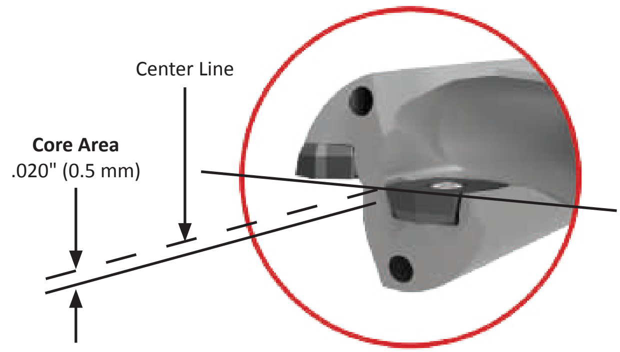
1. Assemble the 4TEX drill into the lathe turret with the top face of the inserts parallel to the x-axis of the machine. This will allow for the diameter offsets to be made using the lathe's x-axis.
2. To increase the nominal diameter, offset the x-axis so the periphery insert moves away from the center of the hole.
3. To decrease the nominal diameter, offset the x-axis so the periphery insert moves toward the center of the hole.

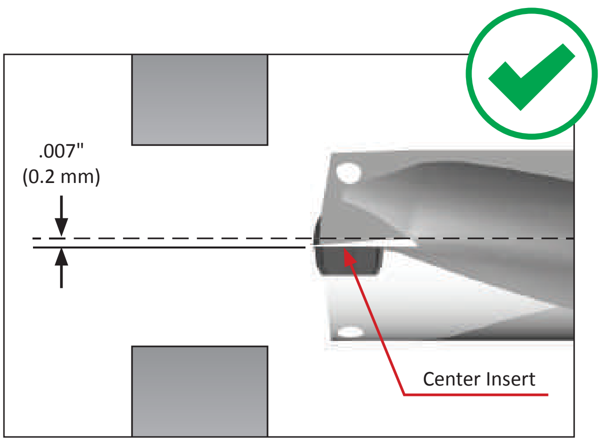
NOTE: Eccentric sleeve is not required when adjusting the diameter of the hole on a lathe.

Center Height Alignment

Proper Center Line Position

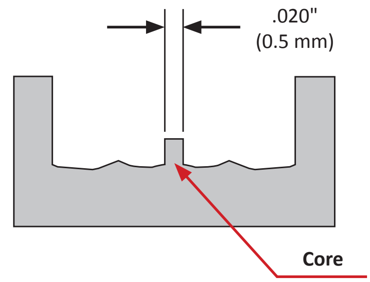


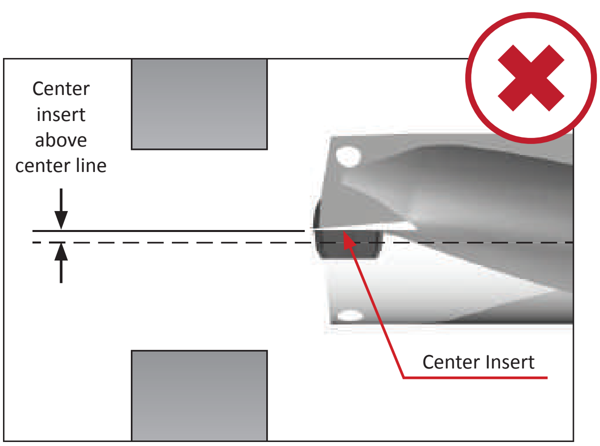




Proper Center Height Alignment

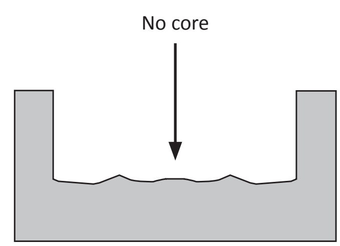
- The correct center height alignment will position the center insert .007" (0.2 mm) below the center line.

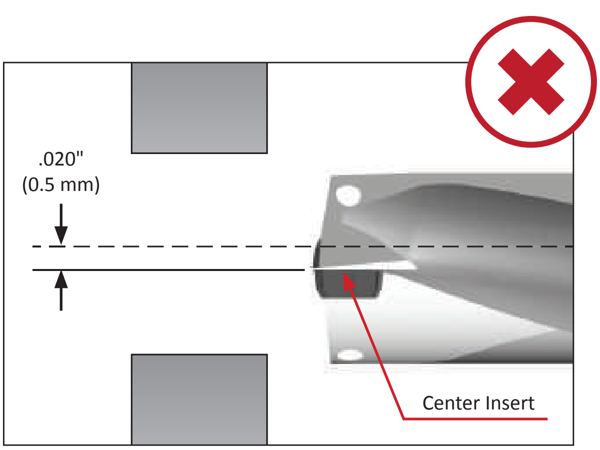




Center Insert Above the Center Line

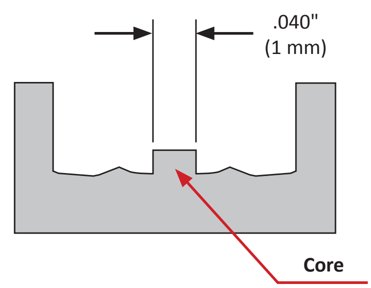
- This will cause fracturing of the center insert
- Requires center height adjustment





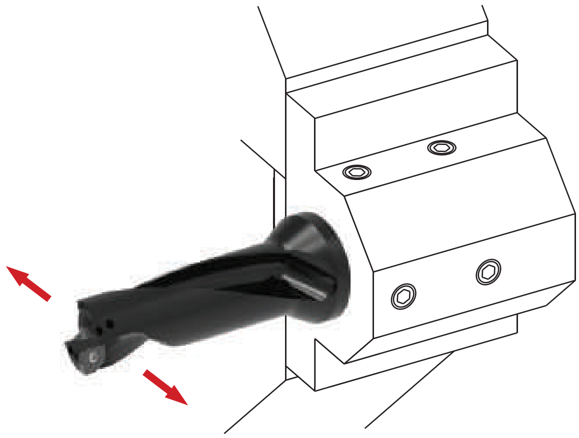
Center Insert Too Far Below Center Line

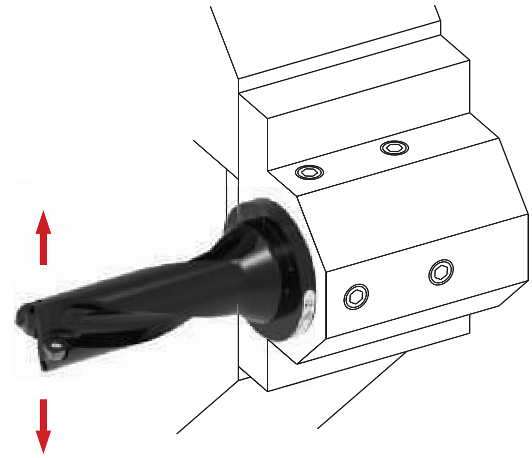
- This will cause the drill to interfere with the drilled hole
- This will impede chip evacuation on the periphery insert
- Requires center height adjustment



Center Height Alignment

How to Correct Issues





Method 1: Adjustment with X-Axis

1. Rotate the drill body so the position of the center line of the inserts is perpendicular to the lathe's x-axis.
2. Use the x-axis to offset the position of the center line in a (+) or (-) direction to increase or decrease the center core diameter at the bottom of the hole.

NOTE: This method does not allow diameter adjustments using the x-axis.

Method 2: Adjustment with Eccentric Sleeve

1. Assemble the drill to the turret using the eccentric sleeve, positioning the center line of the inserts parallel to the x-axis.
2. Align the reference indentation on the drill to the "0" setting on the flange face.
3. Rotate the sleeve (+) or (-) to increase or decrease the center height of the inserts in order to increase or decrease the core diameter at the bottom of the hole.

NOTE: This method still allows diameter adjustments using the x-axis.

NOTE (applies to both methods): Adjusting the center line of the inserts may affect the hole diameter produced. Method two is preferred to make center height adjustments and compensate for hole diameter with the x-axis.

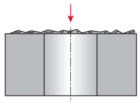
XTP TAP TAS HPU APX 4TX REV OPN SSD ACP BTA WHL CRT ALV BRN THM

Insert Geometry Recommendations

ISO	Material	Hardness (BHN)	Geometry P	Geometry M	Geometry K	Geometry N	Geometry H
P	**Free-Machining Steel** 1118, 1215, 12L14, etc.	100 - 150	○	●			
		150 - 200	●	○			
		200 - 250	●	○			
	Low-Carbon Steel 1010, 1020, 1025, 1522, 1144, etc.	85 - 125	○	●			
		125 - 175	○	●			
		175 - 225	○	●			
		225 - 275	●	○			
	Medium-Carbon Steel 1030, 1040, 1050, 1527, 1140, 1151, etc.	125 - 175	○	●			
		175 - 225	○	●			
		225 - 275	●	○			
		275 - 325	●				○
	Alloy Steel 4140, 5140, 8640, etc.	125 - 175	○	●			
		175 - 225	●	○			
		225 - 275	●				○
		275 - 325	●				○
		325 - 375	○				●
	High-Strength Alloy 4340, 4330V, 300M, etc.	225 - 300	●				
		300 - 350	○				●
		350 - 400	○				●
	Structural Steel A36, A285, A516, etc.	100 - 150	○	●			
		150 - 250	○	●			
		250 - 350	●				○
	Tool Steel H-13, H-21, A-4, 0-2, S-3, etc.	150 - 200	●	○			
		200 - 250	●				○
S	**High-Temp Alloy*** Hastelloy B, Inconel 600, etc.	140 - 220	○	●			
		220 - 310	○	●			
	Titanium Alloy*	140 - 220	○	●			
		220 - 310	○	●			
	Aerospace Alloy* S82	185 - 275	○	●			
		275 - 350	○	●			
M	**Stainless Steel 400 Series** 416, 420, etc.	185 - 275	○	●			
		275 - 350	○	●			
	Stainless Steel 300 Series 304, 316, 17-4PH, etc.	135 - 185	○	●			
		185 - 275	○	●			
	Super Duplex Stainless Steel		○	●			
		135 - 275	○	●			
H	**Wear Plate** Hardox, AR400, T-1, etc.	400	○				●
		500	○				●
		600	○				●
	Hardened Steel	300 - 400	○				●
		400 - 500	○				●
K	**Nodular, Ductile Cast Iron**	120 - 150	●	○			
		150 - 200	●	○			
		200 - 220	●	○			
		220 - 260			●		○
		260 - 320			●		○
	Grey / White Iron	120 - 150			●		○
		150 - 200			●		○
		200 - 220			●		
		220 - 260			●		
		260 - 320			●		
N	**Cast Aluminum**	30				●	
		180				●	
	Wrought Aluminum	30				●	
		180				●	
	Aluminum Bronze	100 - 200	○			●	
		200 - 250	○			●	
	Brass	100	○			●	
	Copper	60				●	

Troubleshooting

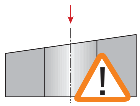
1.



Starting on Uneven Surfaces

- Reduce entry feed by 50% if necessary

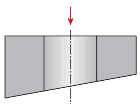
2.



Starting on Angled Surfaces

- Reduce entry feed by 20 - 50%
- Use lower rake geometry if insert chipping occurs

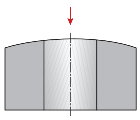
3.



Angled Bore Exit

- Reduce entry feed by 50% on breakout
- Use tough insert and stable corner radius

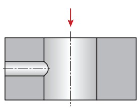
4.



Starting on Convex Surfaces

- Reduce entry feed by 50%
- Use lower rake geometry if insert chipping occurs

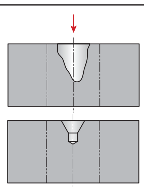
5.



Drilling Through a Cross Hole

- Reduce feed rate 50% if necessary
- Use good coolant flow and monitor chip packing
- Use lower rake geometry if insert chipping occurs

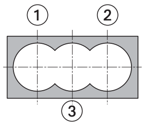
6.



Drilling on a Groove or Large Centering Box

- Reduce entry feed
- Use lower rake geometry for center insert

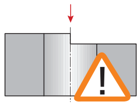
7.



Chain Drilling

- Use good coolant flow
- Reduce feed rate by 50% for interrupted cut
- Use lower rake geometry if insert chipping occurs

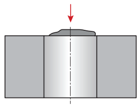
8.



Starting on an Edge

- Reduce entry feed rate by 50%
- Use lower rake geometry if insert chipping occurs

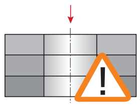
9.



Starting on a Welded Seam

- Reduce entry feed rate by 50%
- Use lower rake geometry if insert chipping occurs

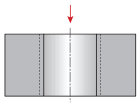
10.



Drilling Through Stacked Plates

- Not recommended

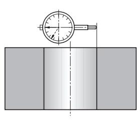
11.



Opening an Existing Hole

- Use flood coolant

12.



Adjustable

- For mills, use eccentric sleeve with end mill holder
- For lathes, use x-axis to adjust offset Ø

NOTE: Refer to maximum offset Ø in data tables

Notes

SECTION

A60

Revolution Drill®

Product Overview

Series	Diameter Range		Length to Diameter Ratio	Shank Options			Inserts per Cartridge	Page
	Imperial (in)	Metric (mm)		Straight	CAT40	CAT50		
34	1.875 - 2.000	47.6 - 50.8	2.2, 3.5, 4.5	✓	✓	✓	2	6 - 7
36	2.000 - 2.200	50.8 - 55.9	2.2, 3.5, 4.5	✓	✓	✓	2	8 - 9
38	2.200 - 2.400	55.9 - 61.0	2.2, 3.5, 4.5	✓	✓	✓	2	10 - 11
42	2.400 - 2.600	61.0 - 66.0	2.2, 3.5, 4.5	✓	✓	✓	2	12 - 13
44	2.600 - 2.800	66.0 - 71.1	2.2, 3.5	✓		✓	3	14
46	2.800 - 3.000	71.1 - 76.2	2.2, 3.5	✓		✓	3	15
48	3.000 - 3.200	76.2 - 81.3	1.0, 2.5	✓		✓	3	16
52	3.200 - 3.400	81.3 - 86.4	1.0, 2.5	✓		✓	3	17
54	3.400 - 3.600	86.4 - 91.4	1.0, 2.5	✓		✓	3	18
56	3.600 - 3.800	91.4 - 96.5	1.0, 2.5	✓		✓	4	19
58	3.800 - 4.000	96.5 - 101.6	1.0, 2.5	✓		✓	4	20

NOTE: Stacked plate styles are also available

Features & Benefits

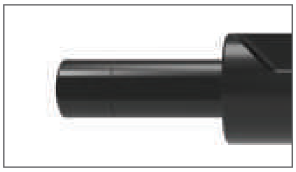
- Adjustability of 0.200" (5.10 mm) on diameter
- Drill depths up to 4.5xD (standard)
- The replaceable cartridges protect your investment
- Adjustable diameter reduces inventory and cost
- The insert design allows for excellent chip control and aggressive penetration rates
- No pilot hole needed

2 Inserts
(34 - 42 series)

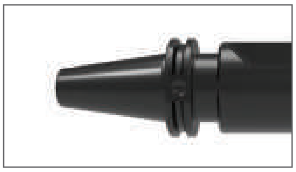
3 Inserts
(44 - 54 series)

4 Inserts
(56 - 58 series)

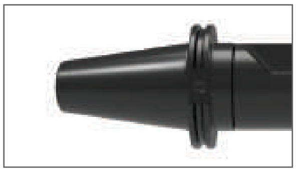
Shank Options



Straight Shank
(all series)



CAT40 Shank
(34, 36, 38, 42 series)

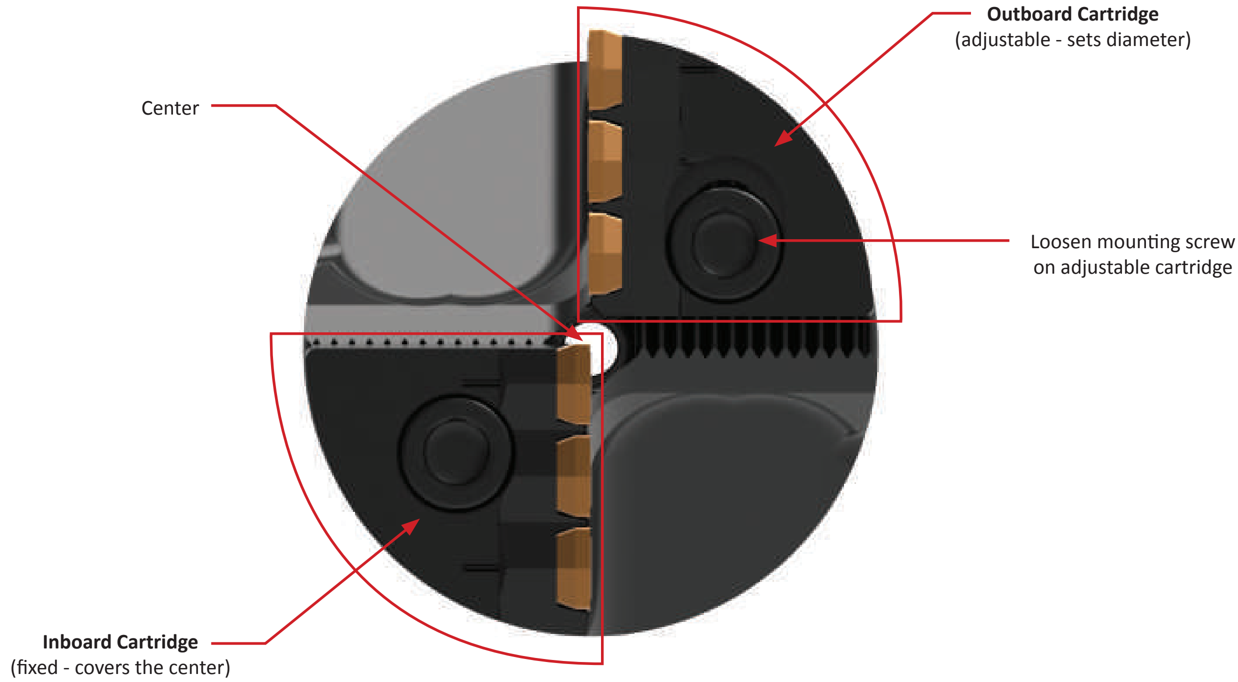


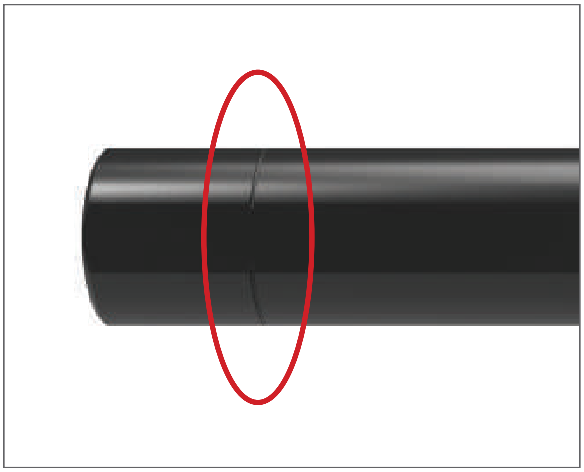
CAT50 Shank
(all series)

Body Lengths

- 1.0xD (48, 52, 54, 56, 58 series)
- 2.2xD (34, 36, 38, 42, 44, 46 series)
- 2.5xD (48, 52, 54, 56, 58 series)
- 3.5xD (34, 36, 38, 42, 44, 46 series)
- 4.5xD (34, 36, 38, 42, 44, 46 series

Product Overview

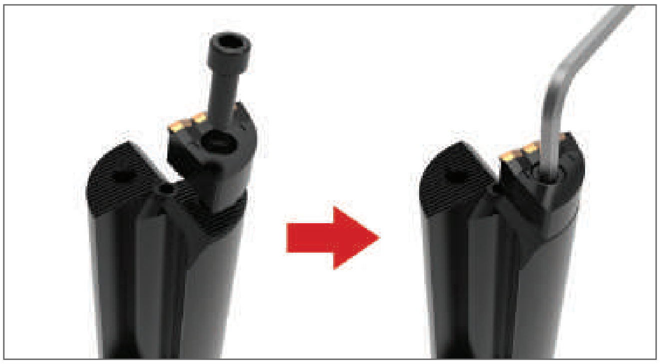




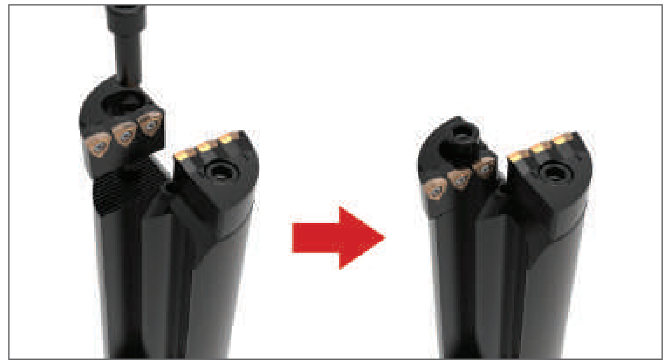
Straight Shanks

- Designed for lathe applications
- Can be cut off for use in end mill holders
- The score mark (circled above) is provided for recommended cut length
- Cut and deburr at the score mark
- This improves rigidity when the body sits against the face of an end mill holder

Set-up Instructions



Step 1:
Mount the fixed cartridge and tighten the mounting screw to 11-14 ft-lbf (15-19 N-m).



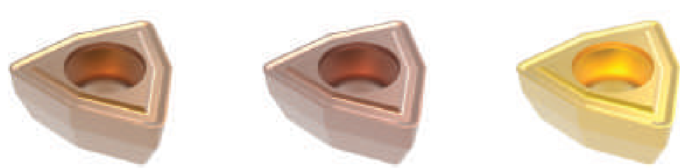
Step 2:
Finger-tighten the mounting screw on the adjustable cartridge.

Step 3:
Set the diameter using the adjustment screw against the mounting screw. Place the drill in a pre-setter to ensure the correct diameter setting.

Step 4:
Tighten the mounting screw to 11-14 ft-lbf (15-19 N-m).

IC Inserts

- The design allows for excellent chip control and aggressive penetration rates
- The proprietary AM200® and AM300® coatings increase tool life above competitors' premium coatings
- The same inserts are used for both Revolution Drill and Opening Drill products



AM300® **AM200®** **TiN**

Insert Application Recommendations

Carbide Grade Options	
C5 (P35)	General purpose carbide grade suitable for most applications. ► *Common application in steels and stainless steels.*
C1 (K35)	Toughest carbide grade. Provides the best combination of edge strength and tool life. ► *Recommended for less rigid applications.*
C2 (K25)	Higher wear resistant carbide suitable for abrasive material applications. ► *Recommended for grey, ductile, and nodular irons.*
Additional Geometry Option	
High Rake (HR)	Provides superior chip control and tool life in long chipping carbon and alloy steels below 200 Bhn.

SECTION

A70

Opening Drill®

Product Overview

Features

- Can be used as a rotating or stationary tool
- Can be used in rough boring operations
- Available in multiple different shanks (see chart below)
- Smooth cutting action and quiet operations in lathes and mills
- Special lengths, diameters, and shanks are available upon request

Advantages

- Opens an existing hole in a single operation
- Ignores core shifts up to 1/8" (3.18 mm) providing straight and true holes without the need for boring
- Allows for large amounts of material removal
- Unique design enables larger holes to be made on low horsepower machines
- Replaceable cartridges protect your investment
- Adjustable diameters reduce inventory and cost

Shank Options


Straight


CAT40


CAT50


BT40


BT50

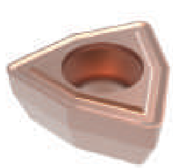

HSK63


HSK100


DIN50


ABS63


AM300®


AM200®


TiN


2 Inserts
(OP1 - OP3 series)


3 Inserts
(OP4 series)

Insert Application Recommendations

Carbide Grade Options	
C5 (P35)	General purpose carbide grade suitable for most applications. ▶ *Common application in steels and stainless steels.*
C1 (K35)	Toughest carbide grade. Provides the best combination of edge strength and tool life. ▶ *Recommended for less rigid applications.*
C2 (K25)	Higher wear resistant carbide suitable for abrasive material applications. ▶ *Recommended for grey, ductile, and nodular irons.*
Additional Geometry Option	
High Rake (HR)	Provides superior chip control and tool life in long chipping carbon and alloy steels below 200 Bhn.

IC Inserts

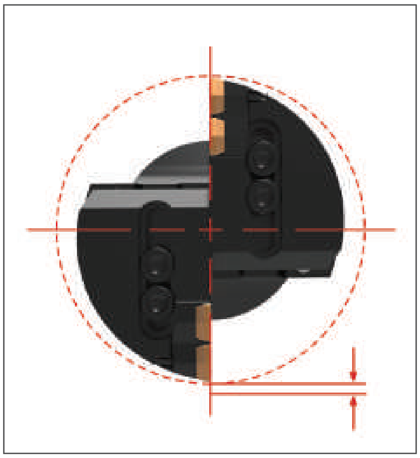
- The design allows for excellent chip control and aggressive penetration rates
- The proprietary AM200® and AM300® coatings increase tool life above competitors' premium coatings
- The same inserts are used for both Revolution Drill and Opening Drill products

Set-up Instructions

Step 1:
Loosen the mounting screws on both cartridges.

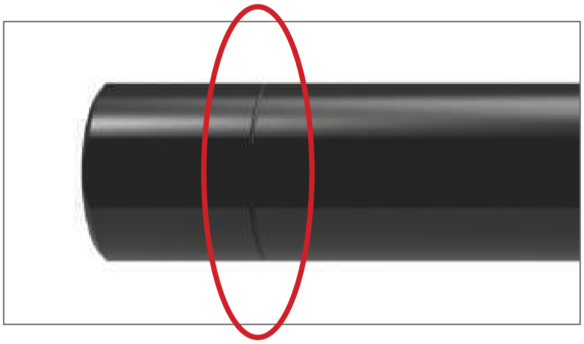
Step 2:
Set one cartridge to the finish diameter by tightening the adjustment screw against the adjustment pin.

Step 3:
Tighten the mounting screws on the cartridge to 11-14 ft-lbf (15-19 N-m).



Step 4:
Set the opposing cartridge with 0.160" to 0.200" radial offset inward by tightening the adjustment screw against the adjustment pin (optimum situation for each insert to remove equal material).

Step 5:
Tighten the mounting screws on the cartridge to 11-14 ft-lbf (15-19 N-m).



Straight Shanks

- Designed for lathe applications
- Can be cut off for use in end-mill holders
- The score mark (circled to the left) is provided for recommended cut length
- Cut and deburr at the score mark
- This improves rigidity when the body sits against the face of an end-mill holder

Notes

SECTION

A91

Structural Steel Solutions

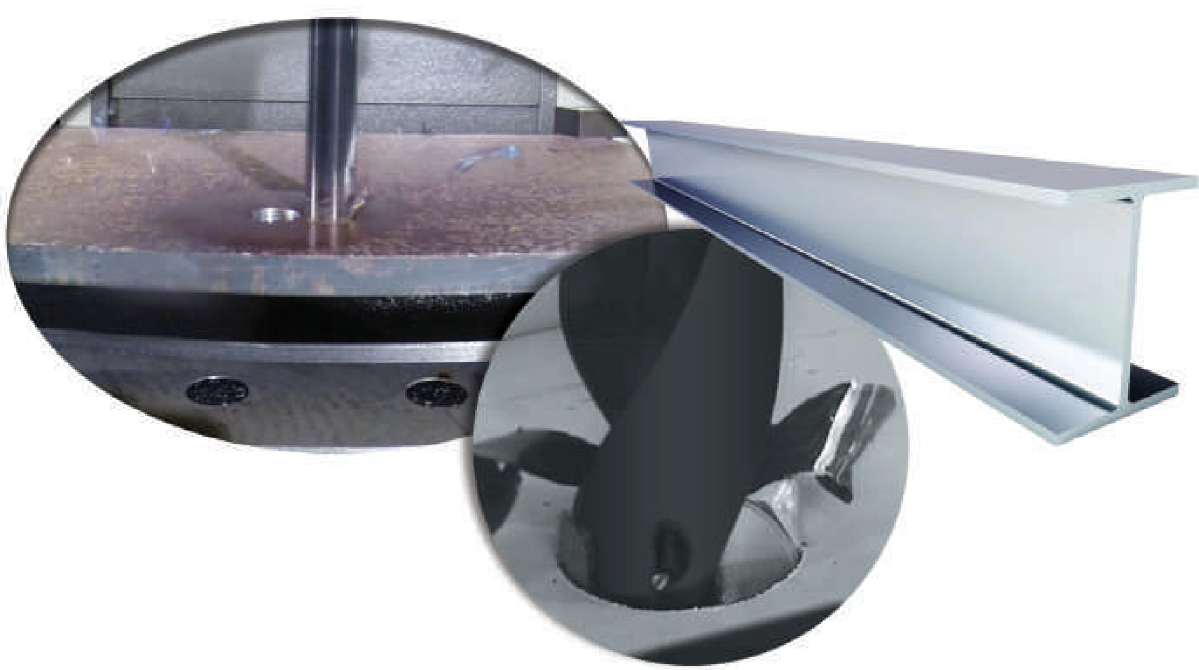
Structural Steel Drilling

Achieving Optimal Results in Structural Steel

Drilling in structural steel materials can be a difficult process, and achieving optimal results becomes a major issue. Allied Machine's structural steel drilling solutions have been specifically designed to produce the best results in the toughest materials. With solutions in both the T-A® and GEN3SYS® XT Pro product lines, you have multiple options to solve your application problems.

Insert Style Comparison

	 GEN3SYS® XT Pro Structural Steel	 T-A® Thin Wall	 T-A® Notch Point®	 T-A® 150° Structural Steel	 T-A® GEN2 High Efficiency
High penetration	✓				
Material less than 7/16" thick		✓			
Material over 7/16" thick	✓		✓	✓	✓
Reduced exit burr			✓	✓	
Includes Notch Point® geometry			✓	✓	✓
Available from carbide	✓				✓
Stocked in common sizes for the Structural Steel industry	✓	✓	✓	✓	✓



Case Study Example

CASE STUDY

Project Profile:	Structural Steel I-Beam Construction
Tooling Solution:	T-A® Structural Steel Drilling System

The Problem:

Previously, the customer was using a competitor spade drill running at the following parameters:

- 650 RPM
- 0.010 IPR (0.25 mm/rev)
- 6.5 IPM (165.1 mm/min)

The tool drilled a 0.875" (22.23 mm) diameter hole to a 0.4375" (11.11 mm) depth. The drill had a tool life of **only 20 holes**.

The poor tool performance was brought to the attention of the technician, who was familiar with Allied Machine products. The following day, Allied Machine tooling was brought in for testing. The customer needed improvement in the tool life of the inserts.

The Solution:

Allied Machine recommended the T-A Structural Steel Drilling System.

- **Insert** = 151A-0028-TW (#1 series T-A insert with TiAlN coating and Thin Wall geometry)
- **Holder** = 25010H-004IS052 (#1 series T-A holder with #4 Morse Taper shank and helical flute)

The tool ran at the following parameters:

- 440 RPM
- 0.010 IPR (0.25 mm/rev)
- 4.4 IPM (111.7 mm/min)

The tool achieved the desired diameter and depth. But most of all, the tool produced **1,500 holes**.

Summary:

The customer was able to take advantage of Allied Machine's vast experience in the structural steel drilling niche. Allied's wide variety of stocked solutions for specific customer problems allows for a remarkable increase in tool life.

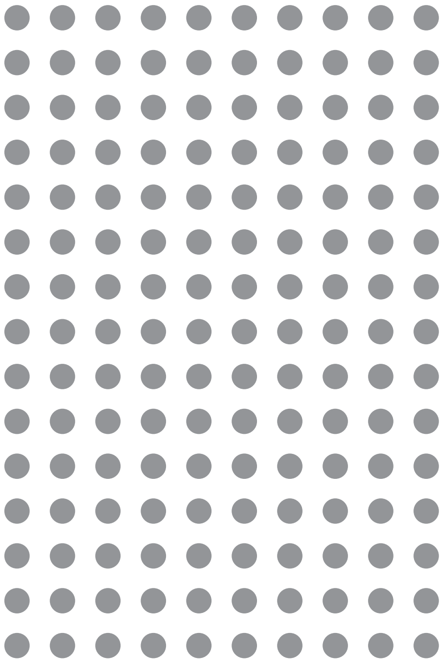
The T-A Structural Steel Drilling System defeated the competition, decreasing the total cost per hole from $2.02 to just $0.22. This reduction resulted in a **savings of 89%** for the customer.

The **PROOF** is in the

NUMBERS

Competitor Insert Tool Life
(number of holes = 20)

T-A Structural Steel Insert Tool Life
(number of holes = 1,500)

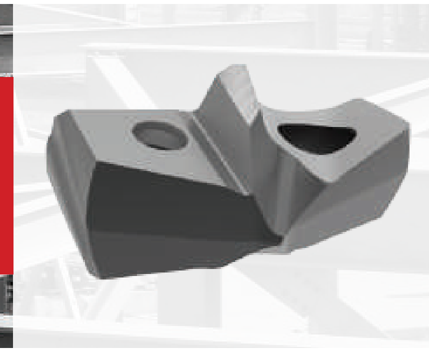


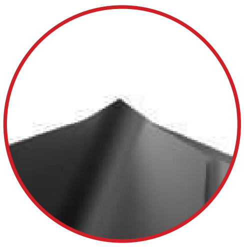
Overall ***SAVINGS*** of

89%

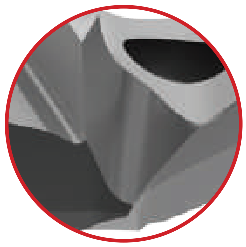
GEN3SYS® XT Pro Structural Steel Drilling System

GEN3SYS® XT Pro *ST*
STRUCTURAL STEEL ENHANCEMENTS

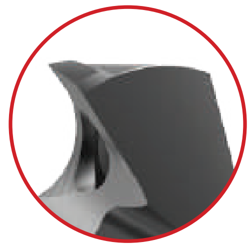




New Point Design
Increases stability without hindering penetration



Redesigned Insert
Provides consistent performance and adds durability



Improved Geometry
Extends tool life and increases insert strength without increasing horsepower consumption

AM420 Coating
Increases heat threshold and extends tool life

Get the Consistency You Need

The challenge of drilling structural steel materials is about to get easier. Developed through a rigorous and thorough testing process, the modified and improved XTST insert is a product of innovation.

Achieve the ***consistent performance*** you need while matching or even exceeding your current parameters.

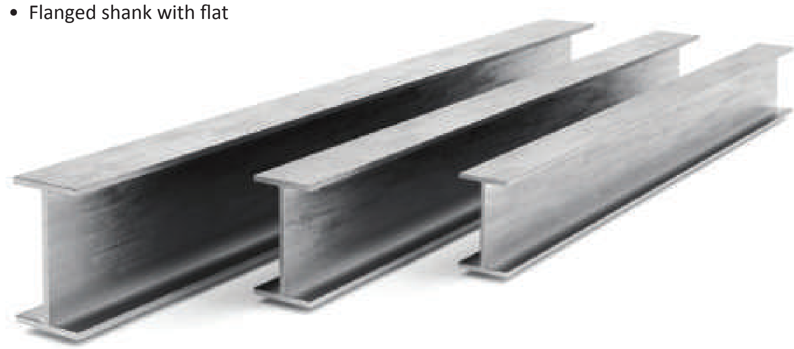
Tough Drilling is Tough No More

Structural steel applications can prove to be difficult to machine, so you need a drill that's been put through the fire to ensure it can conquer those challenging applications.

Rigorous testing and countless hours of design and programming make the XT Pro structural steel insert the optimal drill for structural steel applications.

- Diameter range: 0.4724" - 1.3780" (12.00 mm - 35.00 mm)
- Holders available in 1.5xD, 3xD, 5xD, and 7xD lengths
- Flanged shank with flat

1.5xD 3xD 5xD 7xD



NOTICE: Structural Steel GEN3SYS holders are specifically designed to be used only with XTST geometry inserts. Using other GEN3SYS XT or XT Pro insert geometries in these holders could lead to chip packing and tool failure. Contact Application Engineering for questions regarding proper use of tools.

XTP TAS HPU APX 4TX REV OPN SSD ACP BTA WHL CRT ALV BRN THM

T-A® Structural Steel Drilling System

STRUCTURAL STEEL
ENHANCEMENTS
T-A & T-A GEN2

T-A GEN2 Insert

Available in AM300® Coating

High Efficiency (-HE)

- Improves performance
- Improves tool life
- Improves chip formation in structural steel materials

T-A Inserts

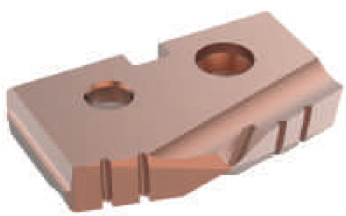
Available in AM200® and TiAlN Coatings

Thin Wall (-TW)

- Designed for drilling 7/16" thick or less I-Beam or structural materials
- Increases hole diameter tolerance
- Improves hole roundness
- Decreases material deflection

Notch Point® (-NP)

- Provides excellent self-centering characteristics
- Reduces bellmouth and tool lead-off
- Reduces axial thrust requirements



Structural Steel (-SS)

- Designed for drilling 7/16" thick or thicker I-Beam or structural materials
- Reduces exit burrs
- Increases stability
- Lowers drilling forces
- Includes Notch Point® web geometry

Holder Anatomy

1. Morse Taper Shank
2. Coolant Inlet
3. Flute (straight or helical)
4. Built-up Body Diameter
5. Coolant Outlets

Straight Flute

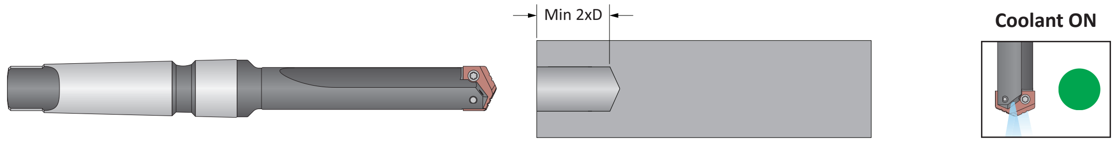
Helical Flute

Deep Hole Drilling Guidelines

For Use with Drills Greater than 9xD (Extended, Long, XL, 3XL, and Special Length)

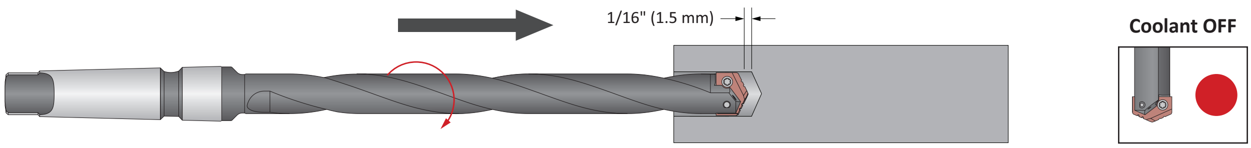
1. Pilot Hole
100 % RPM
100% IPR (mm/rev)

Establish the pilot hole using the same diameter short drill to a depth of 2xD minimum.
Utilize a pilot drill with the same or larger included point angle.



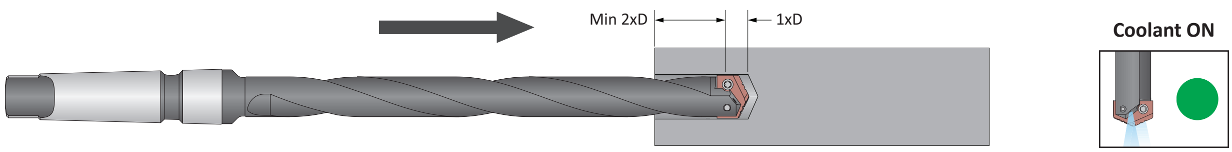
2. Feed-in
50 RPM max
12 IPM (300 mm/min)

Feed the longer drill within 1/16" (1.5mm) short of the established pilot hole bottom at a **maximum of 50 RPM** and 12 IPM (300 mm/min) feed rate.



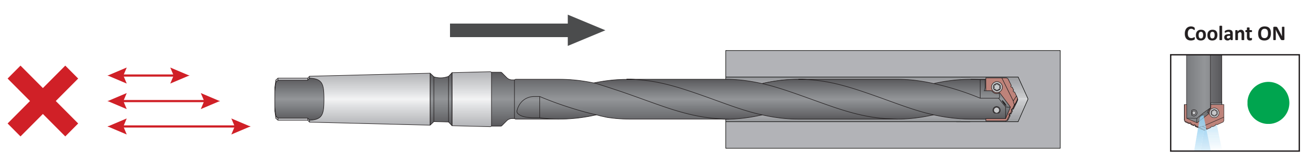
3. Deep Hole Transition Drilling
50 % RPM
75% IPR (mm/rev)

Drill additional 1xD past the bottom of the pilot hole at 50% reduction of recommended speed and 25% reduction of recommended feed.
Minimum of 1 second dwell is required to meet full speed before feeding.



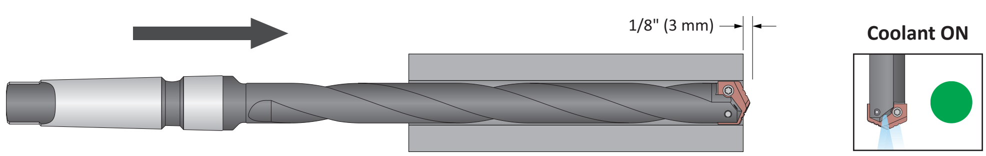
4. Deep Hole Drilling - Blind
100% RPM
100% IPR (mm/rev)

Drill to full depth at recommended speed and feed for longer drill according to Allied speed and feed charts.
No peck cycle recommended.



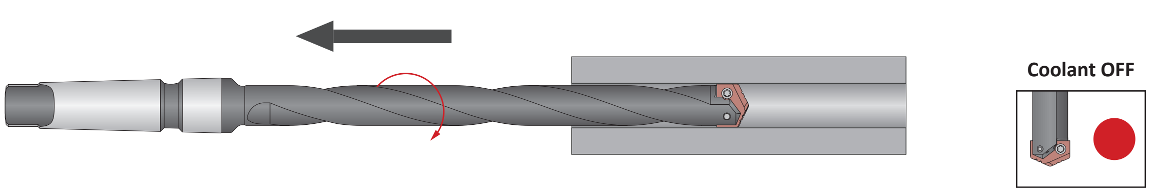
5. Deep Hole Drilling - at Breakout
50% RPM
75% IPR (mm/rev)

For through holes only:
Reduce speed by 50% and feed by 25% prior to breakout.
Do not break out more than 1/8" (3mm) past the full diameter of the drill.



6. Drill Retract
50 RPM max

Reduce speed to a **maximum of 50 RPM** before retracting from the hole.



WARNING **Tool failure can cause serious injury. To prevent:**

- When using holders without support bushing, use a short T-A® holder to establish an initial hole that is a minimum of 2 diameters deep.
- Do not rotate tool holders more than 50 RPM unless it is engaged with the workpiece or fixture.

Visit **www.alliedmachine.com/DeepHoleGuidelines** for the most up-to-date information and procedures. Factory technical assistance is available for your specific applications through our Application Engineering Team.

SECTION

A92

AccuPort 432®

Product Overview

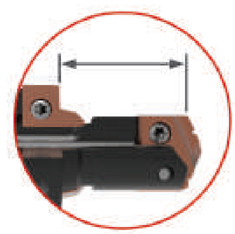
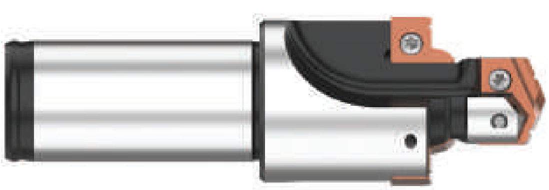
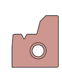
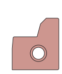
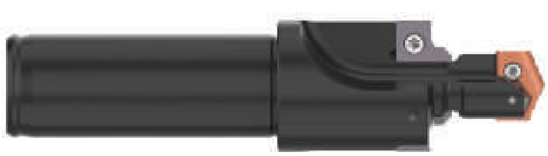
ONE TOOL | **FOUR** OPERATIONS

Advanced Solutions, Outstanding Results

As designers and manufacturing engineers push the limits of production technology to improve productivity and performance, Allied Machine has continued to innovate and develop new solutions like the unique AccuPort 432 hydraulic port contour cutter system. Every product in the AccuPort system is designed to deliver maximum performance in a diverse range of hydraulic port cutting applications and demanding manufacturing environments.

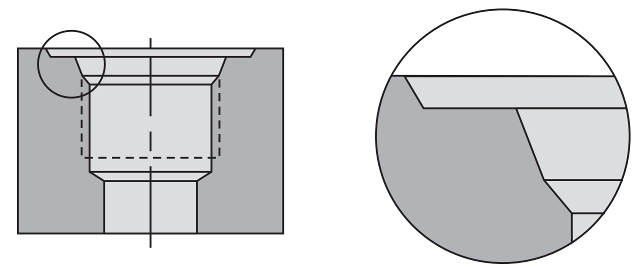
Using precision replaceable inserts for both the drilling and port forming operations, AccuPort eliminates the need for tool regrinding and enables absolute repeatability, excellent surface finish, and reduced cost per hole. The AccuPort drills, forms, and precision-finishes the hydraulic port in **one** pass. This replaces up to three separate cutting operations in a single tool to deliver outstanding improvements in productivity, accuracy, and repeatability.

Hydraulic systems are present in an incredibly diverse range of industries. Anywhere a hydraulic port is required, AccuPort can provide a more cost-effective and higher performance solution in a fraction of the time taken for traditional methods using separate drills, special forming tools, and spot facers.

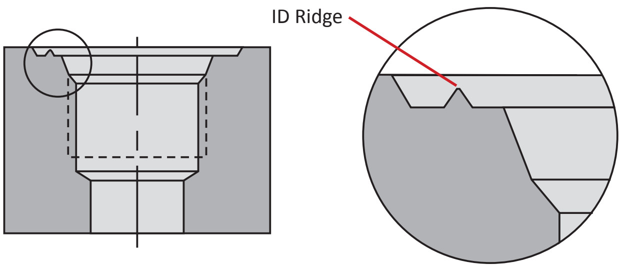
Port Specification		Notes	
Imperial SAE J-1926 ISO 11926-1 MS-16142		Extended minor diameter length option also available	
Metric ISO 6149-1:2006 SAE J-2244/1		Holders made with ID ridge Utilizes inserts with or without ID ridge  ID ridge  No ID ridge	
Military SAE AS5202		Also conforms to AND10050 specification by using an alternate tap drill size for a UN thread	
John Deere® JDS-G173.1		Adheres to John Deere port standards	

Choosing the Right System

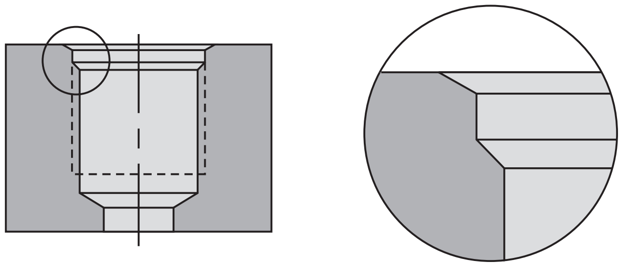
Every product in the AccuPort 432 product line is designed to deliver maximum performance in a diverse range of hydraulic port cutting applications and demanding manufacturing environments. The innovative design delivers the best possible range of benefits in terms of productivity, cost per hole, and tool life.



Imperial J-1926-1



Metric ISO 6149-1: 2006



Military SAE AS5202

Common Industry Sectors and Components

Aerospace
Pumps
Landing Gear
Brake Cylinders
Manifolds

Agriculture
Pumps
Manifolds
Cylinders and Rams
Gear Pumps

Automotive
Motor Valves
Relief Valves
Brake Cylinders
Power Steering Pumps

Marine / Shipbuilding
Pumps
Cylinders and Rams
Motors
Manifolds

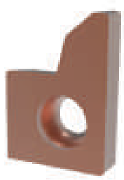
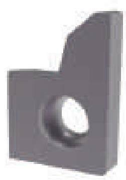
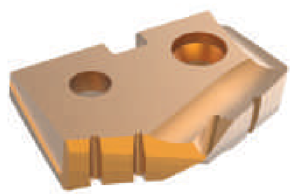
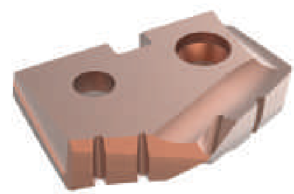
The Complete Package

Producing fully finished threaded hydraulic ports has never been easier. The Port and Thread Finishing Kit includes the AccuPort 432 contour cutter with a dedicated AccuThread® solid carbide thread mill in a single kit. You also receive the T-A® inserts and port form inserts needed to complete the assembly.

Port kits incorporate the AccuThread solid carbide thread mills to increase the manufacturing flexibility by allowing hydraulic ports to be produced in just two operations. In addition, where a unique port profile is required, Allied Machine provides a dedicated special tooling solution using our extensive tool design and manufacturing experience to meet precise specifications.

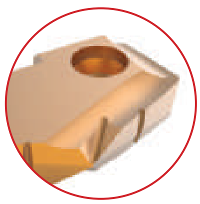
Replaceable Inserts Overview

T-A® Drill Insert Grades			
HSS Super Cobalt (T-A® / T-A® GEN2) Suited for good to rigid machining applications, used for drilling exotic and high alloy materials, or general use when surface speed needs to be increased for use in material hardness up to 350 BHN 121kg.	**Carbide C5 (P40)** (T-A® only) Excellent for drilling free machining steel, low/medium carbon steels, alloy steels, high strength steels, tool steels, and hardened steels.	**Carbide C1 (K10)** (T-A® GEN2 only) Excellent for drilling free machining steel, low/medium carbon steels, alloy steels, high strength steels, tool steels, and hardened steels.	**Carbide C3 (K35)** (T-A® only) Designed for drilling grey/white cast irons. The special geometry offers substantial increases in penetration rates and provides exceptional edge strength and tool life.

Port Form Inserts		T-A GEN2 Inserts		T-A Inserts
				
AM200®	TiAlN	AM300®	AM200®	TiN

T-A GEN2 Standard Geometry

- Designed for rigid machining applications, primarily used for drilling exotic and high alloy materials
- Ideal for general use when the surface speed needs to be increased



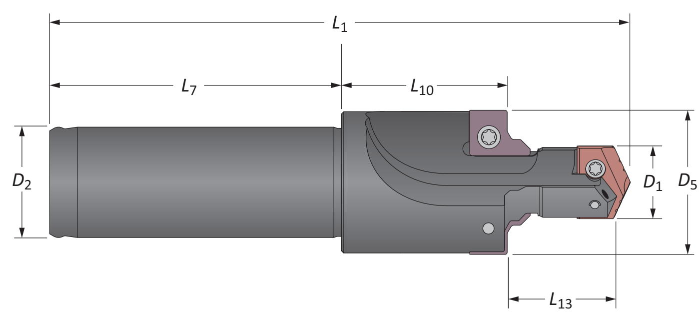
T-A Standard Geometry

- First choice for machining aluminum
- Enhanced geometry improves chip formation and hole quality
- TiN coating improves heat resistance and extends tool life

Made-to-Order Tool Specifications

Scan and email a copy of the table below to Allied's Application Engineering Department to receive pricing for a made-to-order AccuPort 432 Port Contour Cutter.

Send emails to appeng@alliedmachine.com



Tube Dash No.	Specification	Port Thread Size	D_1	L_{13}	D_5	L_{10}	L_1	D_2	L_7
	☐ J1926 ☐ ISO 6149 ☐ ISO 6149 (no ridge) ☐ JDS-G173.1 ☐ AS5202								

Company Name ______

Contact Name ______

Phone ______

Distributor Name ______

Fax ______

SECTION

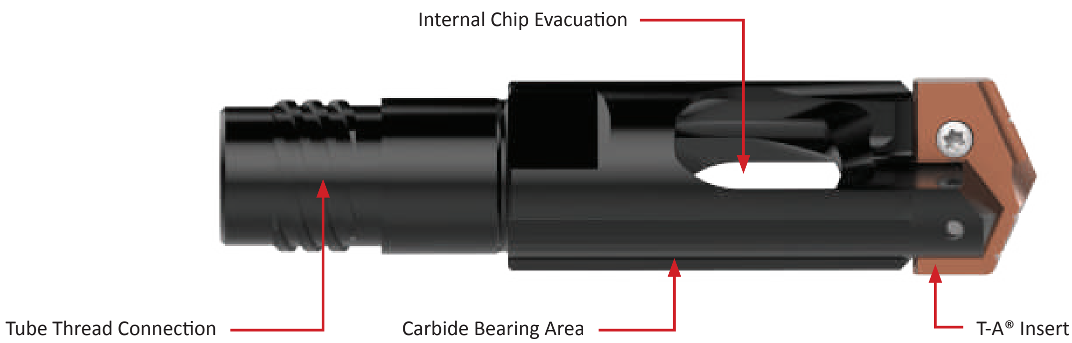
A93

BT-A Drill

System Overview

BTA Machining

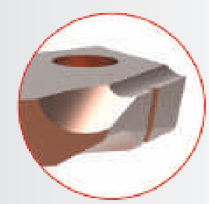
BTA machining is the reverse of typical gun drilling systems. The BT-A drill is a drill head consisting of a holder body and a replaceable tip T-A® insert. The drill head threads into an STS (single tube system) cylindrical tube with a diameter smaller than the drill head. The difference in diameter forms an annular area between the hole and the tube OD. This allows high-volume coolant to be directed to the cutting edge.



- ***Improve hole straightness***
  with the laser clad bearing area
- ***Eliminate the need for resharpening***
  with replaceable cutting edges
- ***Reduce your inventory***
  with the replaceable T-A® feature
- ***Compatibility***
  heads are compatible with standard BTA-STS systems
- ***Balanced cutting forces***
- ***Patented design***

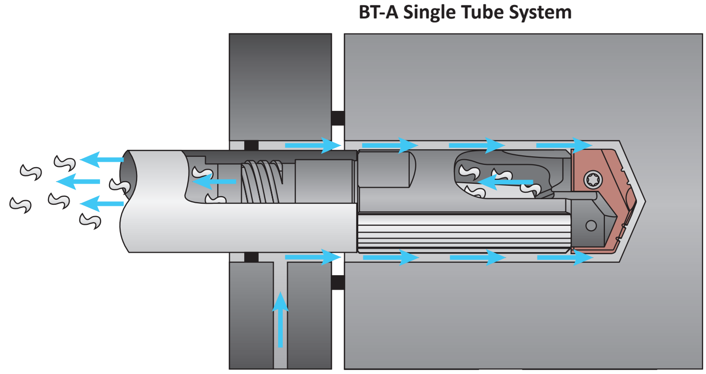
T-A Insert: BT-A Geometry (-BT)

- Low thrust web geometry reduces Z-axis requirements
- Tiny chip (-TC) lip geometry improves chip formation
- Polished cutting surface eliminates material buildup

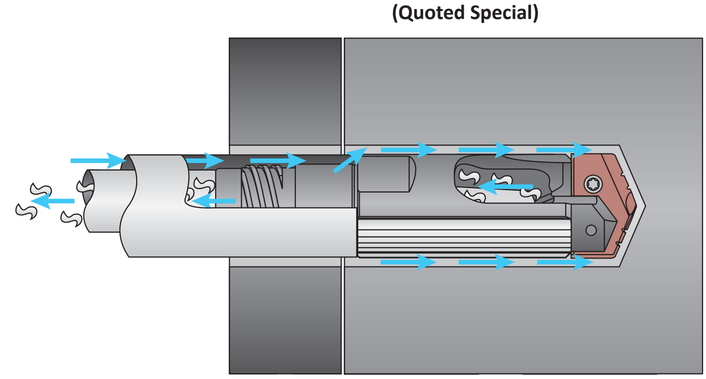


2x *INCREASE* in penetration rates over traditional BTA heads

BT-A Single Tube System



BT-A Double Tube (Ejector) System (Quoted Special)



SECTION

B10

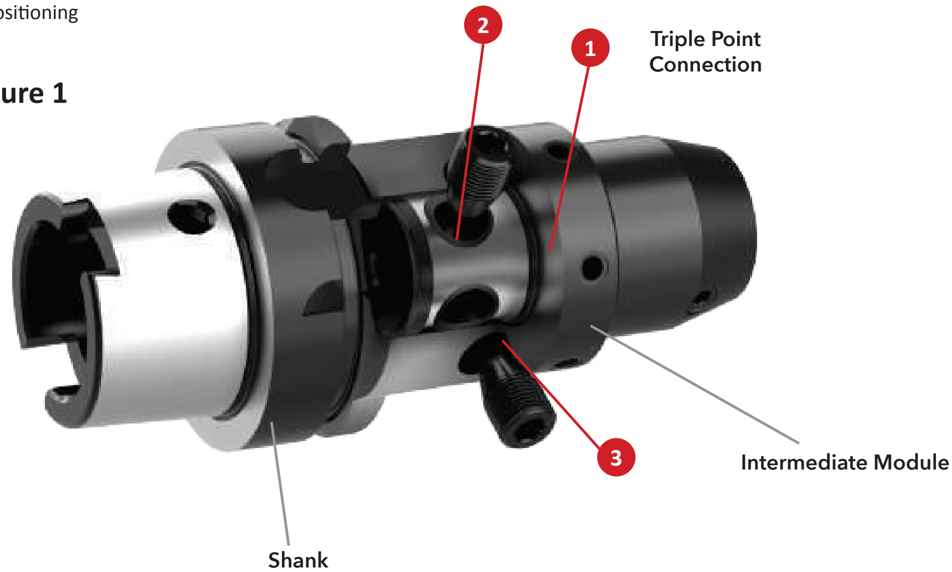
Wohlhaupter® MultiBore System Tools

The MVS Connection

Wohlhaupter® GmbH developed the world's first modular tooling system called MultiBore® in 1973. The modular tooling system was designed to be compatible with all machine spindles, and its connection features a triple point clamp (Figure 1). The triple point clamp (1, 2, & 3) maximizes rigidity and creates an accuracy less than 3μm when the tool is changed.

- Ensures high axial clamping forces on mating surfaces
- Provides maximum rigidity
- Threaded taper pins for precise cutter positioning

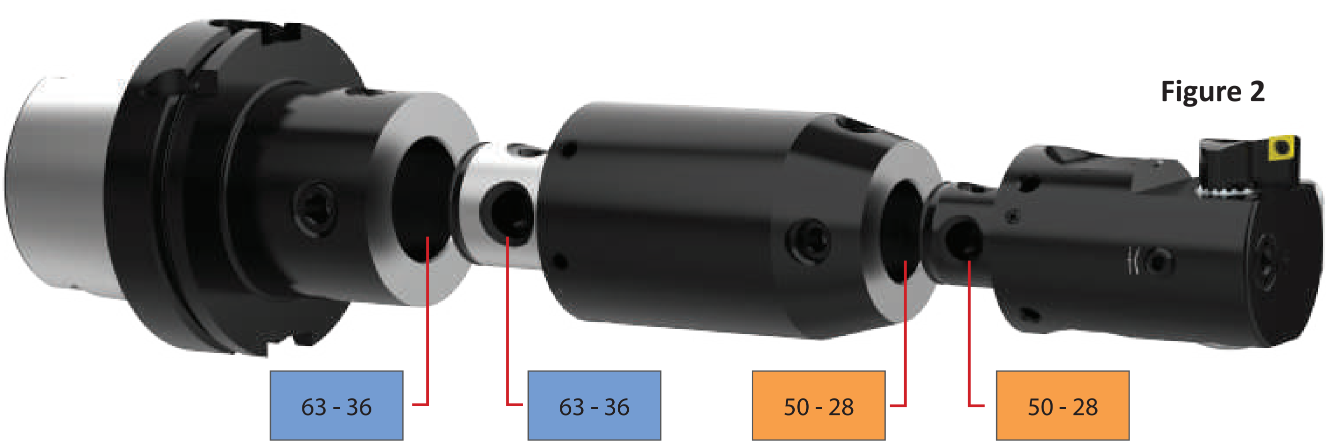
Figure 1



MVS Connection Color Guide

Wohlhaupter® created a unique color-coding system to find the right connections for different tool components quickly and easily. Each Wohlhaupter / MVS connection size has its own color-coding. Simply match the colors to select the correct combination of tool components (Figure 2). We've also incorporated the color-coding system into our packaging to reduce setup time even more.

Figure 2



M 8	19.5 - 22	22 - 11	25 - 14	32 - 18	40 - 22	50 - 28	63 - 36	80 - 36	100 - 56	D 40	D 60

MVS Connection Example - Selecting the Correct Intermediate Module

Example Machining Parameters

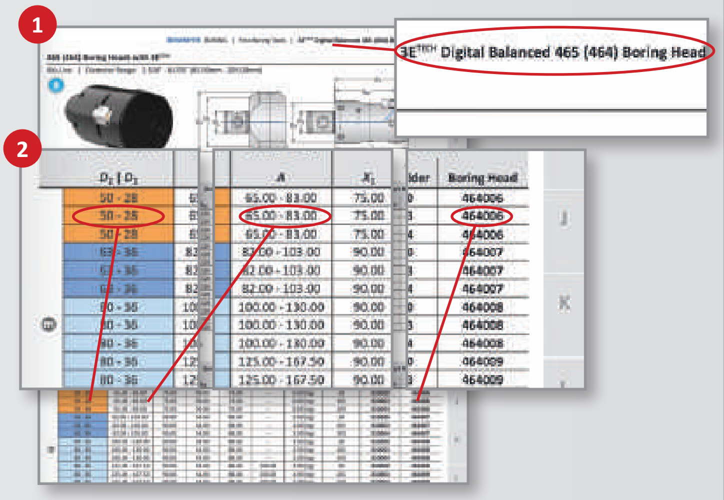
Hole Diameter: 2.942" (75.00mm)
Hole Depth: 7.874" (200.00mm)
Machine Spindle: HSK-A 100 DIN 69893

1 Select your boring tool

- Example: 464 Balanced Boring Head

2 Select the tool that meets the required boring range

- MVS connection: 50 - 28
- *A* = 2.559" - 3.268" (65.00mm - 83.00mm)
- Part No. : **464006**

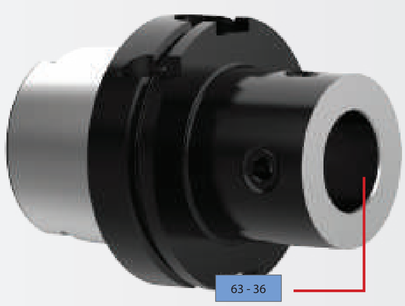


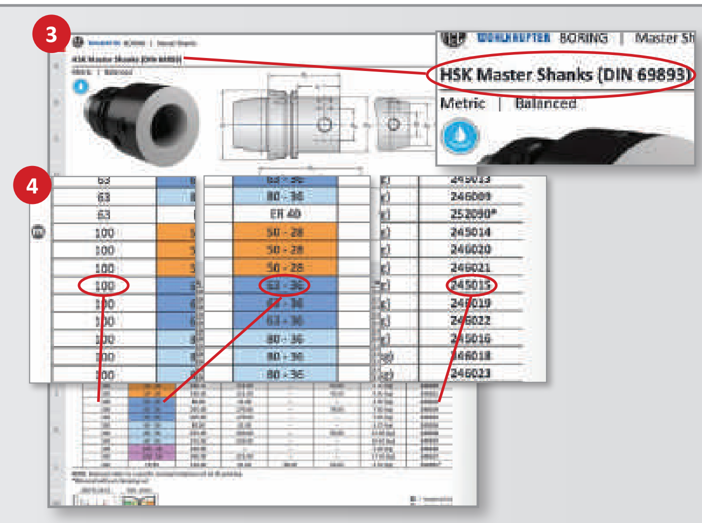
3 Select the master shank that fits your spindle

- Machine spindle: DIN 69893 HSK

4 Choose the connection that fits your application

- MVS connection: 63 - 36
- Part No. : **245015**

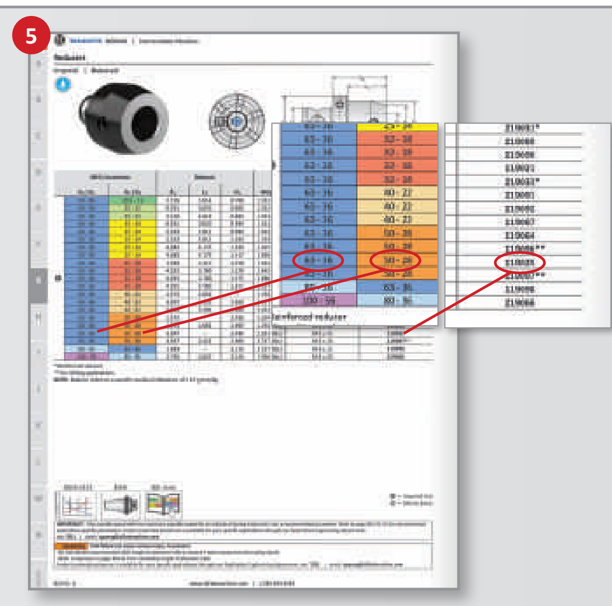




5 Select the correct intermediate module

Must meet the hole depth requirements and MVS connections for both the boring head and shank

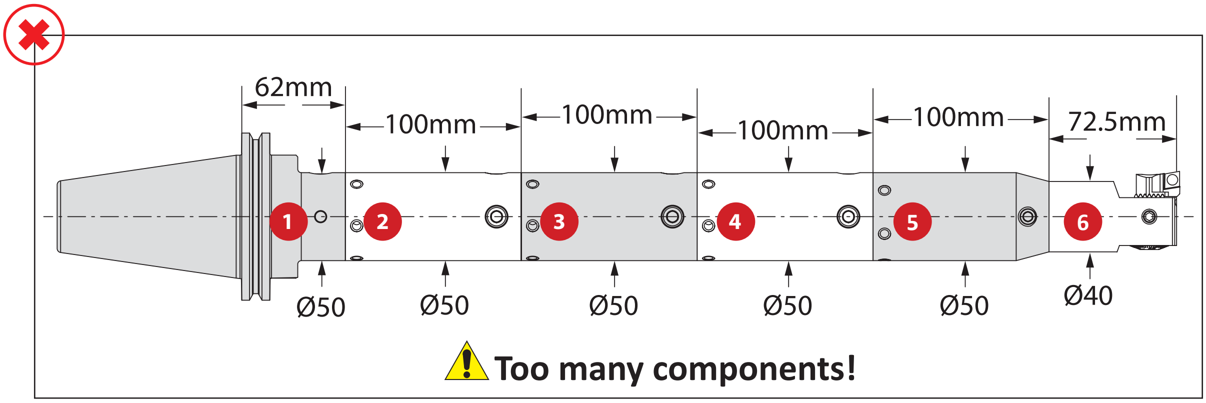
Hole depth: 7.874" (200.00mm)
Boring tool 464006 length (X_1): 2.953" (75.00mm)
Master shank 245015 length (L_1): 2.008" (51.00mm)
Minimum required length of adapter: 2.913" (74.00mm)
Boring tool 464006 MVS connection: 50 - 28
Master shank 245015 MVS connection: 63 - 36
Adapter Part No.: **119025**



XTP TAP TAS HPU APX 4TX REV OPN SSD ACP BTA WHL CRT ALV BRN THM

Guidelines for not Exceeding Recommended Length to Diameter Ratio

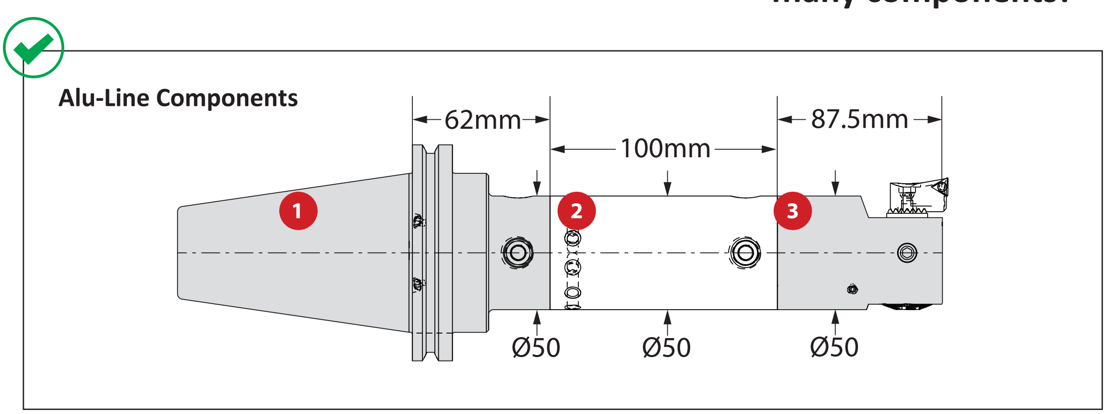
To calculate, see graphics below:



NOTE: Length-to-diameter ratio is calculated using body diameters not cutting diameter.
NOTE: Do not exceed recommended 10xD length-to-diameter ratio or exceed four total components (including shank).

Step 1: Find L : D by component	**Step 2:** Add each L : D Average
1 1.2 = 62/50	1.2
2 2.0 = 100/50	2.0
3 2.0 = 100/50	2.0
4 2.0 = 100/50	2.0
5 2.0 = 100/50	2.0
6 1.8 = 72.5/40	+ 1.8
	11.0 = L : D ratio

Too long with too many components!



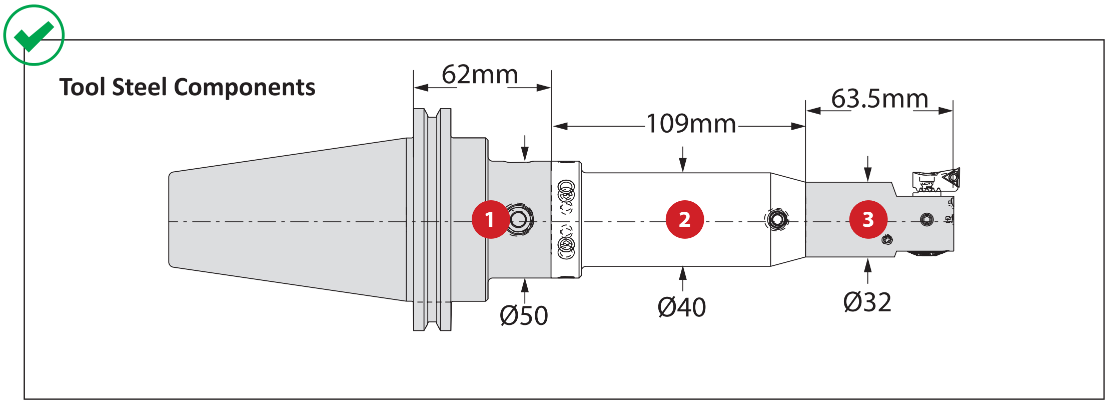
NOTE: Length-to-diameter ratio is calculated using body diameters not cutting diameter.
NOTE: Do not exceed recommended 5xD length-to-diameter ratio when using Alu-Line (Aluminum) components or exceed four total components (including shank).

Step 1: Find L : D by component	**Step 2:** Add each L : D average
1 1.2 = 62/50	1.2
2 2.0 = 100/50	2.0
3 1.8 = 87.5/50	+ 1.8
	5.0 = L : D ratio

WARNING **Tool failure can cause serious injury. To prevent:**
-Do not exceed recommended 10xD length-to-diameter ratio or exceed four total components (including shank)
-When using Alu-Line® components, do not exceed recommended 5xD length-to-diameter ratio
-When using tool steel components, do not exceed recommended 6xD length-to-diameter ratio
-When using a heavy metal component, do not exceed recommended 8xD length-to-diameter ratio
-When using a carbide shank, do not exceed recommended 9xD length-to-diameter ratio
-When using a NOVI[TECH]® module, do not exceed recommended 10xD length-to-diameter ratio
Factory technical assistance is available for your specific applications through our Application Engineering department. *ext:* **7611** | *email:* **appeng@alliedmachine.com**

Guidelines for not Exceeding Recommended Length to Diameter Ratio

To calculate, see graphics below:



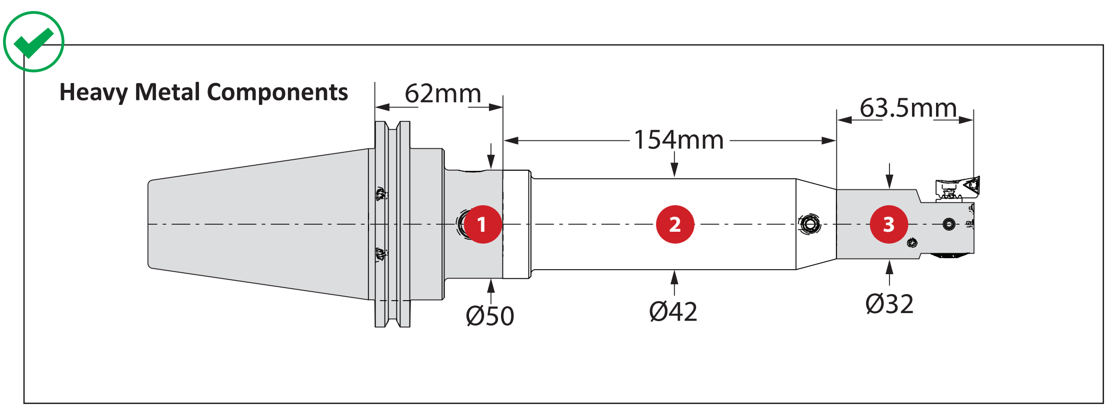
NOTE: Length-to-diameter ratio is calculated using body diameters not cutting diameter.
NOTE: When using steel components, do not exceed recommended 6xD length-to-diameter ratio or exceed four total components (including shank).

Step 1: Find L : D by component

1. 1.2 = 62/50
2. 2.7 = 109/40
3. 2.0= 63.5/32

Step 2: Add each L : D average

1.2
2.7
+ 2.0
5.9 = L : D ratio



NOTE: Length-to-diameter ratio is calculated using body diameters not cutting diameter.
NOTE: When using a heavy metal components, do not exceed recommended 8xD length-to-diameter ratio or exceed four total components (including shank).

Step 1: Find L : D by component

1. 1.2 = 62/50
2. 3.6 = 154/42
3. 2.0 = 63.5/32

Step 2: Add each L : D average

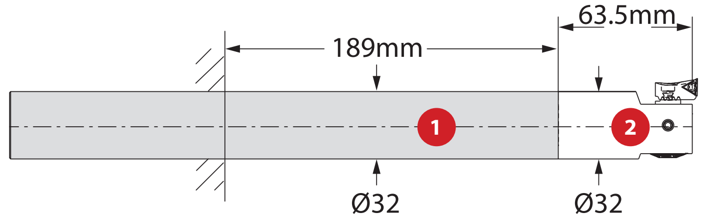
1.2
3.6
+ 2.0
6.8= L : D ratio

⚠ WARNING **Tool failure can cause serious injury. To prevent:**
-Do not exceed recommended 10xD length-to-diameter ratio or exceed four total components (including shank)
-When using Alu-Line® components, do not exceed recommended 5xD length-to-diameter ratio
-When using tool steel components, do not exceed recommended 6xD length-to-diameter ratio
-When using a heavy metal component, do not exceed recommended 8xD length-to-diameter ratio
-When using a carbide shank, do not exceed recommended 9xD length-to-diameter ratio
-When using a NOVI^TECH® module, do not exceed recommended 10xD length-to-diameter ratio
Factory technical assistance is available for your specific applications through our Application Engineering department. *ext:* **7611** | *email:* **appeng@alliedmachine.com**

Guidelines for not Exceeding Recommended Length to Diameter Ratio

To calculate, see graphics below:

Carbide Shank

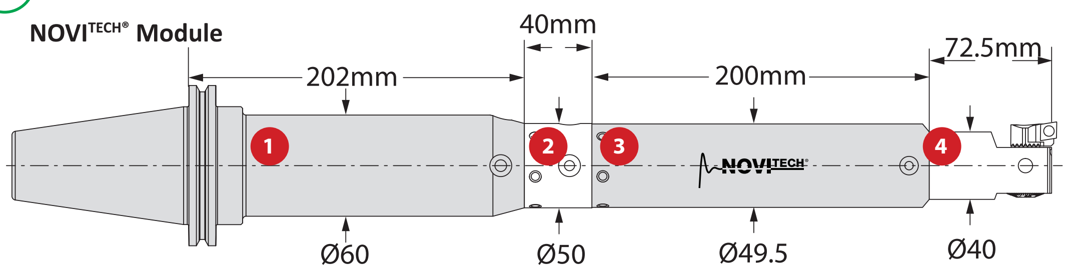


NOTE: Length-to-diameter ratio is calculated using body diameters not cutting diameter.
NOTE: When using carbide shank components, do not exceed recommended 9xD length-to-diameter ratio or exceed four total components.

Step 1: Find L : D by component
1. 8.1 = 189/32
2. 2.0 = 63.5/32

Step 2: Add each L : D average
5.9
+ 2.0
7.9 = L : D ratio

NOVI[TECH]® Module



NOTE: Length-to-diameter ratio is calculated using body diameters not cutting diameter.
NOTE: Do not exceed recommended 10xD length-to-diameter ratio when using NOVI[TECH] intermediate modules or exceed four total components (including shank).
NOTE: The NOVI[TECH] intermediate module should always be assembled as close as possible to the cutting edge (i.e. the next component behind the boring head).

Step 1: Find L : D by component
1. 3.2 = 202/60
2. 0.8 = 40/50
3. 4.0 = 200/49.5
4. 1.8 = 72.5/40

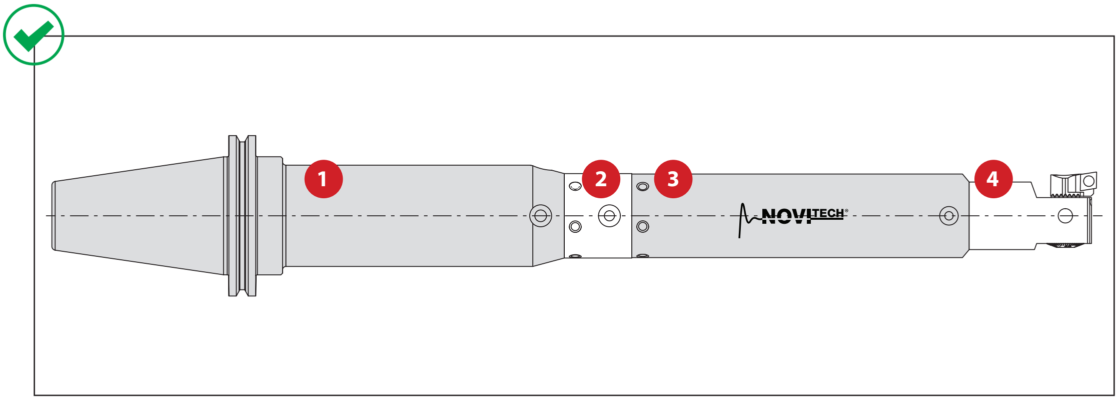
Step 2: Add each L : D average
3.2
0.8
4.0
+ 1.8
9.8 = L : D ratio

Component	Length to Diameter Ratio
Alu-Line	5xD
Tool Steel Components	6xD
Heavy Metal	8xD
Carbide	9xD
NOVI[TECH]®	10xD

⚠ WARNING **Tool failure can cause serious injury. To prevent:**
-Do not exceed recommended 10xD length-to-diameter ratio or exceed four total components (including shank)
-When using Alu-Line® components, do not exceed recommended 5xD length-to-diameter ratio
-When using tool steel components, do not exceed recommended 6xD length-to-diameter ratio
-When using a heavy metal component, do not exceed recommended 8xD length-to-diameter ratio
-When using a carbide shank, do not exceed recommended 9xD length-to-diameter ratio
-When using a NOVI[TECH]® module, do not exceed recommended 10xD length-to-diameter ratio
Factory technical assistance is available for your specific applications through our Application Engineering department. *ext:* **7611** | *email:* **appeng@alliedmachine.com**

Calculating Tool Assembly Weight

To calculate, see graphics below:



Step 1: Find weight for each component circled in the example table below

Example:

	MVS Connection	Boring Range	4 Boring Head					
	D_1 & D_2	A	X_1	X_2	L_2	D_5	Weight	Part No.
i	40 - 22	2.087 - 2.598	2.953	1.535	2.854	–	1.543 (lbs)	**320004**
m	40 - 22	53.01 - 65.98	75.00	39.00	72.50	–	0.70 (kg)	**320004**

Step 2: Calculate total assembly weight

1 6.6 kg
2 0.6 kg
3 3.5 kg
+ 4 0.7 kg

11.4 kg

Step 3: Consult machine tool builder to ensure tool assembly weight does not exceed machine capabilities.

⚠ WARNING **Exceeding weight capacity for machine tool spindle and tool changer can cause machine damage and/or serious injury. To prevent:**
-Consult machine tool builder for machine's weight limitations.
Factory technical assistance is also available for specific applications through our Application Engineering department. *ext:* **7611** | *email:* **appeng@alliedmachine.com**

BORING SOLUTIONS *YOU NEED*

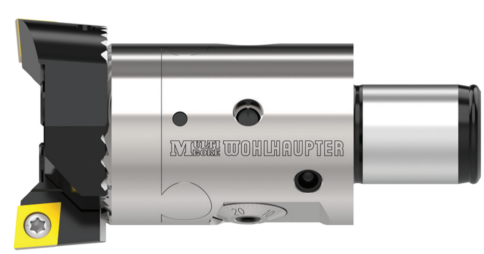
THE
COMPLETE RANGE

Ø 0.016" - 128.15 " (0.4mm - 3255mm)

Standard

Balance

Digital Balance



Combi-Line

Plus **many** *more*

SECTION

B20

Criterion® Boring Systems

Setup Instructions | Standard Adjusting Boring Heads

Adjusting Standard Adjusting Boring Heads (see figure B1)

1. Loosen locking screw (6).
2. Turn dial screw (3) to desired graduation.
3. Tighten locking screw (6) to proper torque spec (laser marked on tool).

IMPORTANT: Do not loosen the gib screws (5). It can cause poor performance.

NOTE: To machine smaller bore diameters, turn dial screw (3) counterclockwise one full rotation to remove any backlash. Once backlash is mitigated, turn dial screw (3) clockwise to desired graduation.

No.	Part
1	Bar holder
2	Boring head body
3	Dial screw
4	Bar holder set screws
5	Gib screws (DO NOT ADJUST)
6	Locking screw

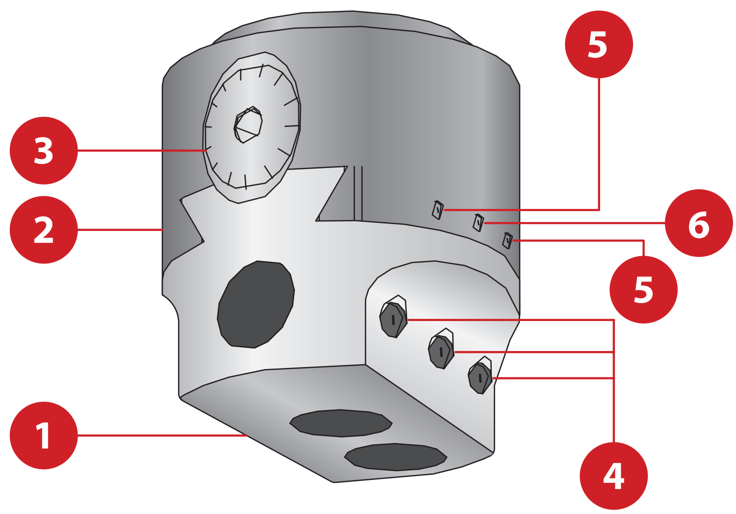


Figure B1

Setup Instructions | Micro Adjusting Boring Heads

Setting Up Micro Adjusting Boring Heads (see figure B2)

Set the microadjusting dial screw range

1. The microadjusting dial screws (4) only have a total range of 0.006" (0.152 mm) on diameter. To zero, turn dial (4) clockwise until dial screw bottoms out. Turn the dial (4) two complete turns counterclockwise. Turn dial (4) one half turn clockwise. Dial is now centered for 0.003" (0.076 mm) positive or negative travel.

Setting the diameter of the boring head

2. Loosen locking screw (6).
3. Turn dial screw (3) to adjust to the desired diameter using a presetter or plunge indicator or the dial screw (3).
4. Tighten the locking screw (6) to the proper torque spec (laser marked on the tool).
   - Microadjustments will be made at the machine.
5. Make a shallow test cut (roughly 0.250" deep) to determine the actual diameter.
6. Use the microadjusting dial (4) to adjust to the finish diameter. Do not release the locking screw (6) for microadjustments.
   - If the hole diameter is more than 0.002" from the target hole size return to step two.

IMPORTANT: Do not loosen the gib screws (5). It can cause poor performance.

NOTE: Backlash occurs when the diameter of the boring head needs to be decreased. To remove backlash, turn the dial (3) counterclockwise at least one half of a full rotation past the desired adjustment. Once backlash is mitigated, turn dial screw (3) clockwise to the desired adjustment.

No.	Part
1	Insert holder
2	Boring head body
3	Dial screw
4	Microadjusting dial screw
5	Gib screws (DO NOT ADJUST)
6	Locking screw

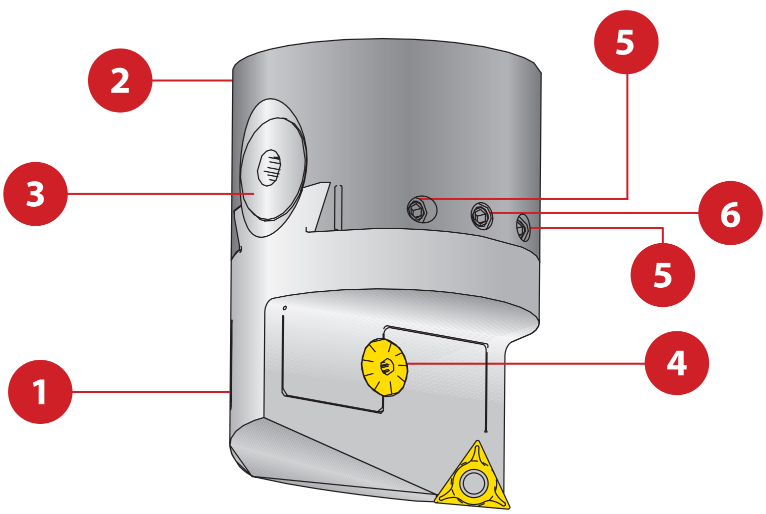
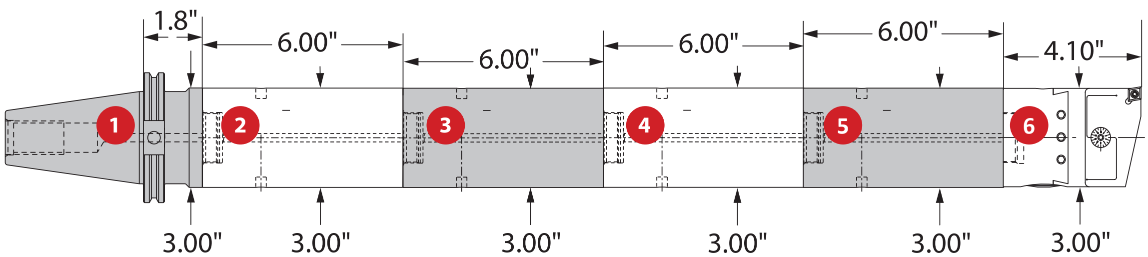


Figure B2

Guidelines for Not Exceeding Recommended Length-to-Diameter Ratio

To calculate, see graphics below:



Too many components!

*Length to diameter ratio is calculated using body diameters, not cutting diameter.

Step 1: Find L : D by component

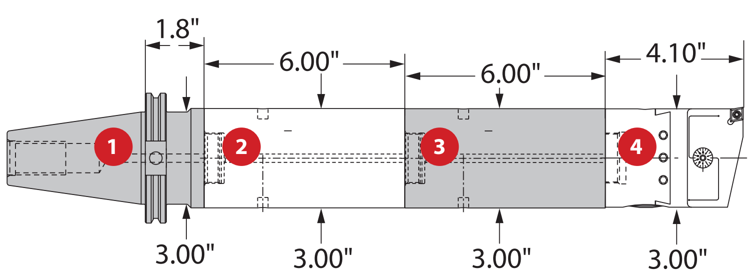
1. 0.6 = 1.88/3.00
2. 2.0 = 6.00/3.00
3. 2.0 = 6.00/3.00
4. 2.0 = 6.00/3.00
5. 2.0 = 6.00/3.00
6. 1.4 = 4.10/3.00

Step 2: Add each L : D Average

0.6
2.0
2.0
2.0
2.0
\+ 1.4

10.0 = L : D ratio

Too Long!



*Length-to-diameter ratio is calculated using body diameters, not cutting diameter.

Step 1: Find L : D by component

1. 0.6 = 1.88/3.00
2. 2.0 = 6.00/3.00
3. 2.0 = 6.00/3.00
4. 1.4 = 4.10/3.00

Step 2: Add each L : D Average

0.6
2.0
2.0
\+ 1.4

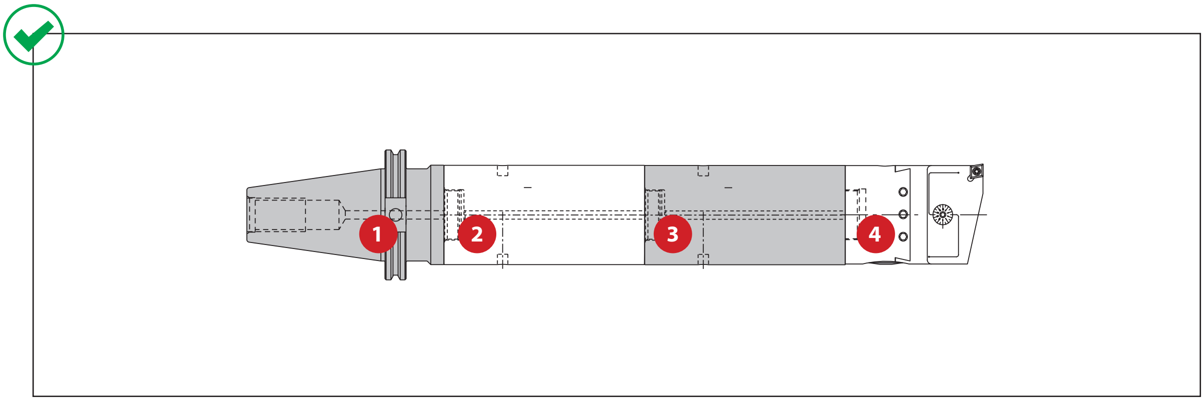
6.0 = L : D ratio

WARNING **Tool failure can cause serious injury. To prevent:**
- Do not exceed recommended 9xD length-to-diameter ratio or exceed 4 total components (including shank)

Factory technical assistance is available for your specific applications through our Application Engineering department. *ext:* **7611** | *email:* **appeng@alliedmachine.com**

Calculating Tool Assembly Weight

To calculate, see graphics below:



Step 1: Find weight for each component

Example:

Boring Range		4 Boring Head				
*D*1	Thread Connection	*L*1	*D*2	Weight	Insert Form	Order Number
1.050 - 1.320	⅞ - 20	2.690	1.000	0.50 (lbs)	CC..215...	**CB1000CC**
1.050 - 1.320	⅞ - 20	2.690	1.000	0.50 (lbs)	TC..215...	**CB1000TC**
1.300 - 1.600	⅞ - 20	2.900	1.250	0.80 (lbs)	CC..215...	**CB1250CC**
1.300 - 1.600	⅞ - 20	2.900	1.250	0.80 (lbs)	TC..215...	**CB1250TC**
1.585 - 2.700	⅞ - 20	3.200	1.500	1.30 (lbs)	CC..325...	**CB1500CC**
1.585 - 2.700	⅞ - 20	3.200	1.500	1.30 (lbs)	TC..325...	**CB1500TC**
2.060 - 3.320	⅞ - 20	3.590	2.000	2.40 (lbs)	CC..325...	**CB2000CC**
2.060 - 3.320	⅞ - 20	3.590	2.000	2.40 (lbs)	TC..325...	**CB2000TC**
3.065 - 5.065	1½ - 18	4.100	3.000	5.80 (lbs)	CC..325...	**CB3000CC**
3.065 - 5.065	1½ - 18	4.100	3.000	5.80 (lbs)	TC..325...	**CB3000TC**
27.00 - 33.00	⅞ - 20	68.35	25	0.23 (kg)	CC..0602...	**CB025MCC**
27.00 - 33.00	⅞ - 20	68.35	25	0.23 (kg)	TC..1102...	**CB025MTC**
33.00 - 41.00	⅞ - 20	73.65	32	0.36 (kg)	CC..0602...	**CB032MCC**
33.00 - 41.00	⅞ - 20	73.65	32	0.36 (kg)	TC..1102...	**CB032MTC**
41.00 - 68.00	⅞ - 20	81.25	38	0.59 (kg)	CC..09T3...	**CB038MCC**
41.00 - 68.00	⅞ - 20	81.25	38	0.59 (kg)	TC..16T3...	**CB038MTC**
53.00 - 84.00	⅞ - 20	91.30	50	1.09 (kg)	CC..09T3...	**CB050MCC**
53.00 - 84.00	⅞ - 20	91.30	50	1.09 (kg)	TC..16T3...	**CB050MTC**
78.00 - 128.00	1½ - 18	104.25	76	2.36 (kg)	CC..09T3...	**CB076MCC**
78.00 - 128.00	1½ - 18	104.25	76	2.36 (kg)	TC..16T3...	**CB076MTC**

Imperial (in) = 0.00005" adjustment on diameter
Metric (mm) = 0.001 mm adjustment on diameter

Step 2: Calculate total assembly weight

1 8.03 lbs
2 11.50 lbs
3 11.50 lbs
\+ 4 5.80 lbs

36.83 lbs

Step 3: Consult machine tool builder to ensure tool assembly weight does not exceed machine capabilities.

⚠ WARNING **Exceeding weight capacity for machine tool spindle and tool changer can cause machine damage and/or serious injury. To prevent:**
- Consult machine tool builder for machine's weight limitations.

Factory technical assistance is also available for specific applications through our Application Engineering department. *ext:* **7611** | *email:* **appeng@alliedmachine.com**

Notes

SECTION C

ALVAN® Reamers

Case Study Example

CASE STUDY

Project Profile: Grey Cast Iron Hydraulic Transmission Component

Tooling Solution: ALVAN® Reamer - Monobloc Style

The Problem:

Previously, the customer was using a competitor boring tool running at the following parameters:

- 3802 RPM
- 500 SFM
- 0.003 IPR
- 11.41 IPM

With 2 passes, the tool made a 0.5023" diameter hole to a 1.20" depth.

- Cycle time = 12.6 seconds
- Tool life = 75 parts

Seeking to streamline the production process, the customer needed to increase tool life and lower the cost of production.

The Solution:

Allied Machine recommended the ALVAN® monobloc style reamer.

- **Reamer** = 92440 series carbide, uncoated, V lead

The tool ran at the following parameters:

- 2200 RPM
- 289 SFM
- 0.019 IPR
- 41.80 IPM

The tool achieved the desired diameter and depth, and the results achieved the customer's goals.

- Cycle time = 1.7 seconds
- Tool life = 3,176 parts

The Advantages:

The customer was able to lower the cost of production and increase the tool life.

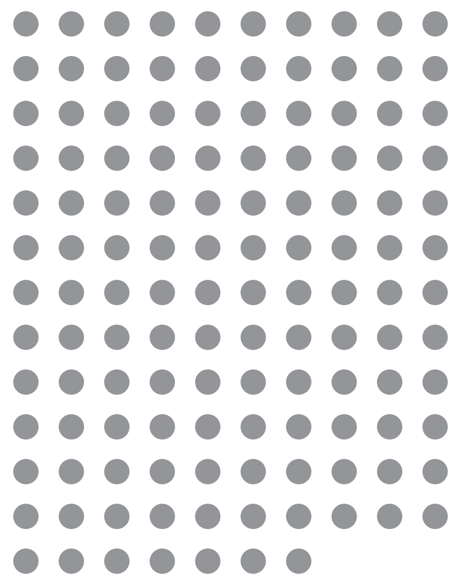
- Reduced cycle time ***from 12.6 seconds to 1.7 seconds***
- Increased tool life ***from 75 parts to an incredible 3,176 parts***
- Total cost savings = **$2,407 (or 52%)**

The **PROOF** is in the

NUMBERS

Tool Life: Competitor Boring
(number of parts = 75)

Tool Life: ALVAN® Monobloc Style Reamer
(number of parts = 3,176)

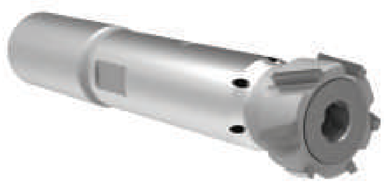


Overall ***SAVINGS*** of

52%

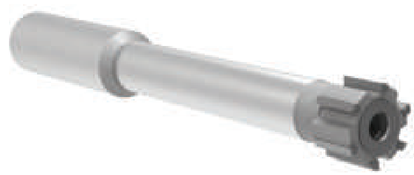
Reaming Overview

REAMER STYLES



Replaceable Head
Pages C: 10 - 19

- Diameter range: 11.80 mm - 60.60 mm
- Heads are available as fixed or expanding for improved productivity
- Straight or left-hand helical flutes provide solutions for both through and blind holes
- Cylindrical or modular shanks improve concentricity



Monobloc
Pages C: 20 - 29

- Diameter range: 5.80 mm - 32.10 mm
- Available with central or radial through coolant
- Can be used for through or blind holes
- Cylindrical shanks improve concentricity
- Expandable to accommodate for wear

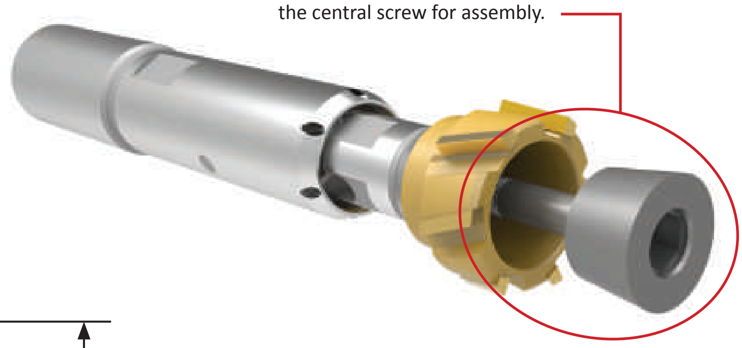
Cutting Ring
Pages C: 30 - 53

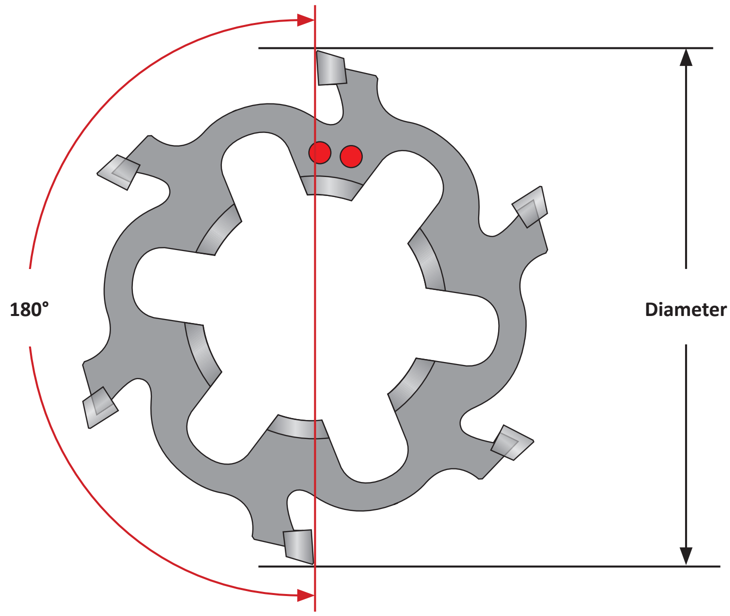
- Diameter range: 17.60 mm - 200.60 mm
- The cutting edges are positioned asymmetrically to assure the best roundness of the hole
- Holes with tight tolerances can be accommodated, and the expansion ensures a perfect holding of the reaming diameter

General Reaming Notes

- If the depth is over 9xD, use a short length reamer to pilot the hole. Then finish with the longer length ⚠.
- For blind hole applications, always use central coolant. If in doubt, contact Allied's Application Engineering department.
- More stock allowance can be taken in softer materials. Less stock allowance should be taken in harder materials.
- A common practice to rapid out of the cut on through holes and to breakout only 2mm past the reaming depth.

IMPORTANT: Always use Molykote® (anti-seize applicant) on the conical seat and the threads on the central screw for assembly.





NOTE: The position of the dimples indicates which two cutting teeth are 180° opposed. Diameter measurements should be taken from these two cutting teeth.

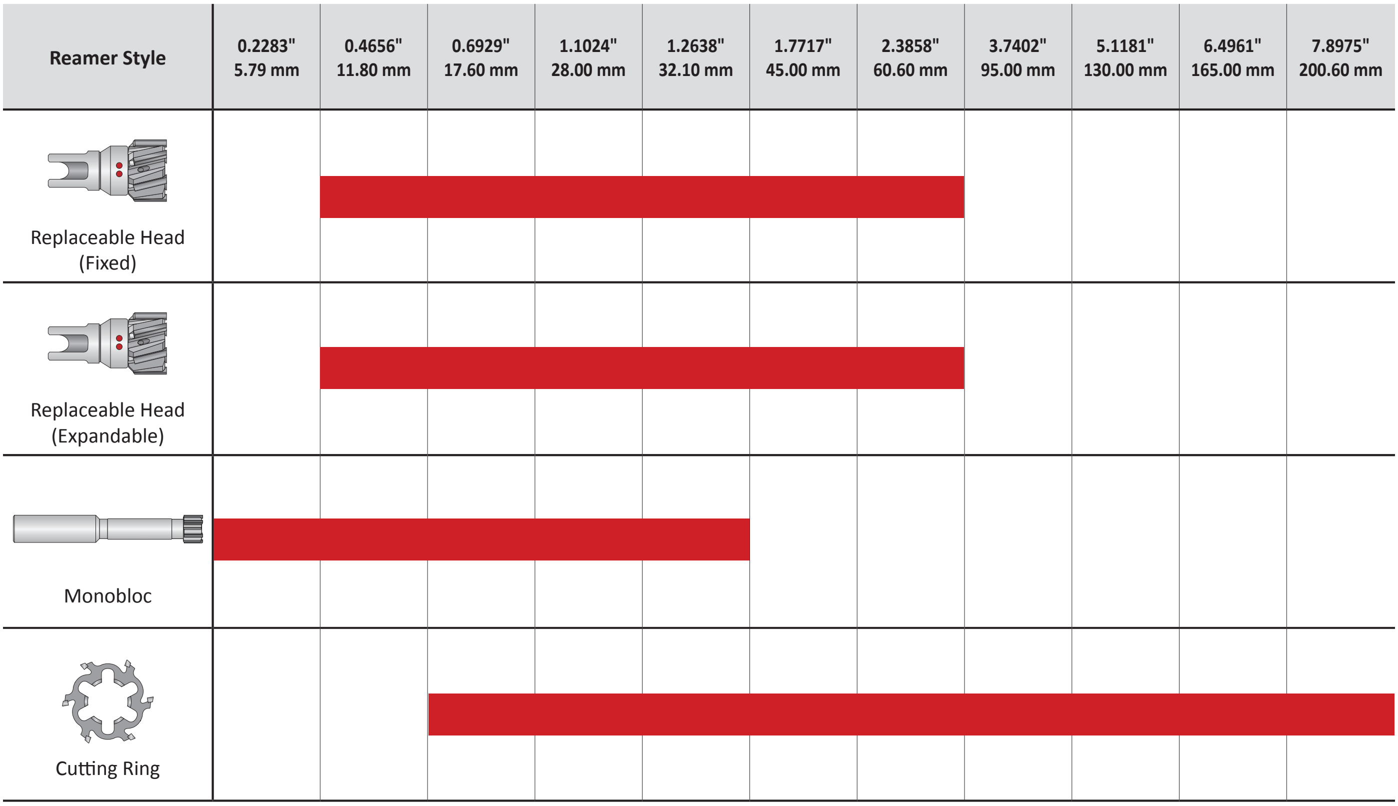
⚠ WARNING **Tool failure can cause serious injury. To prevent:**
- When using holders without support bushing, use a shorter reamer to establish the initial hole diameter that is a minimum of 2 diameters deep.
- Do not rotate reamers more than 50 RPM unless it is engaged with the workpiece or fixture.

Factory technical assistance is available for your specific applications through our Application Engineering Team. *ext:* **7611** | *email:* **appeng@alliedmachine.com**

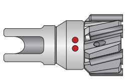
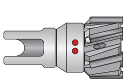
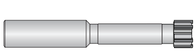
XTP TAP TAS HPU APX 4TX REV OPN SSD ACP BTA WHL CRT ALV BRN THM

Quick Selection Guide

Breakdown by Diameter



Breakdown by Features

Reamer Style	Capable Tolerance	Fastest Setup	Replaceable Cutting Head	Expandable to Adjust for Wear	Recondition Available	Cylindrical Shanks	Modular Shanks	Through-Coolant Options
 Replaceable Head (fixed)	H7	✓	✓		✓	✓	✓	✓
 Replaceable Head (expandable)	H6		✓	✓	✓	✓	✓	✓
 Monobloc	H6			✓	✓	✓		✓
 Cutting Ring	H6		✓	✓	✓	✓	✓	✓

How the Reamer Works

How the Reamer **Works**

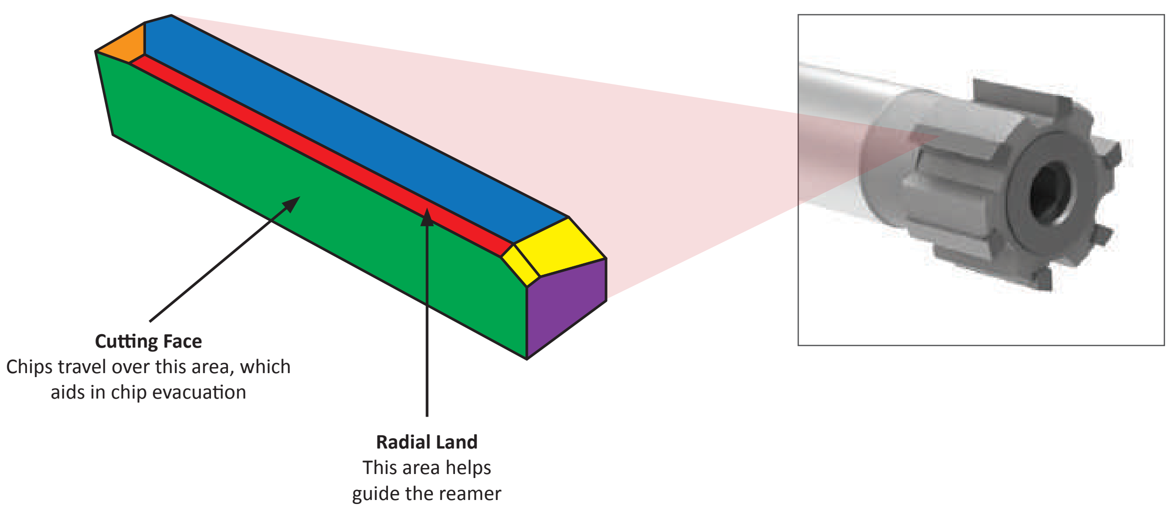
- The cut is made in the lead-in zone (3), and the chip is made on the cutting face (1). The chip is removed by coolant.
- The lead-in (3) is defined depending on the application, the workpiece material, and the stock allowance.
- The radial land (2) is important for holding a good alignment, improving the surface roughness, and giving an effect similar to burnishing. The dimension of the radial land depends on the diameter.
- The radial land (2) is manufactured to be tapered on the rear.
- Fixed reamers are manufactured at the exact tapered value. Expandable reamers must be adjusted to the exact diameter. Both are already supplied at the nominal diameter by the manufacturer.
- The undercut of the cutting edge (5) avoids retract marks on the piece when the reamer is retracted from the cut.
- The front of the cutting edge (6) does not cut; if this feature is needed, a frontal lead must be supplied.

When to **Apply** a Reamer

- When the requested tolerance on diameter is IT8 or less
- When the requested finish is 63 µin (1.6 µmm) Ra or greater
- When the critical geometry characteristics of the hole are the roundness and straightness
- When parts are being mass produced
- When the parts are large and expensive

Elements of the Cutting Tooth

- (1) Cutting Face
- (2) Radial Land
- (3) Lead-in / Primary Face / Secondary Face
- (4) Rear Face
- (5) Undercut of Cutting Edge
- (6) Front of Cutting Edge



XTP TAP TAS HPU APX 4TX REV OPN SSD ACP BTA WHL CRT ALV BRN THM

Reamer Recommendation Guide

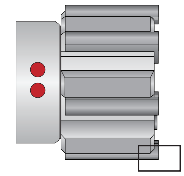
ISO	Material	Hardness (BHN)	Through Hole – Uninterrupted – Lead	Through Hole – Uninterrupted – Substrate & Coating	Through Hole – Interrupted – Lead	Through Hole – Interrupted – Substrate & Coating	Blind Hole – Uninterrupted – Lead	Blind Hole – Uninterrupted – Substrate & Coating	Blind Hole – Interrupted – Lead	Blind Hole – Interrupted – Substrate & Coating
P	**Free Machining Steel** 1118, 1215, 12L14, etc.	Below 150 150 Above	N or E	Cermet Uncoated	E	Cermet Uncoated	K	Cermet Uncoated	V	Cermet Uncoated
P	**Low Carbon Steel** 1010, 1020, 1522, 1144, etc.	Below 250	N or E	Cermet Uncoated	E	Cermet Uncoated	K	Cermet Uncoated	V	Cermet Uncoated
P	**Medium Carbon Steel** 1030, 1040, 1050, 1140, 1151, etc.	Below 300	N or E	Cermet Uncoated	E	Cermet Uncoated	K*	Cermet Uncoated	V	Cermet Uncoated
P	**Alloy Steel** 4140, 5140, 8640, etc.	Below 350	G or M*	Cermet Uncoated	M*	Cermet Uncoated	K*	Cermet Uncoated	G*	Cermet Uncoated
P	**High Strength Alloy** 4340, 4330V, 300M, etc.	240 - 450	G or M*	Carbide Alcrona	M*	Carbide Alcrona	K*	Carbide Alcrona	G*	Carbide Alcrona
P	**Structural Steel**	–	E	Cermet	M	Carbide TiAlN	K	Cermet	G	Carbide TiAlN
P	**Tool Steel**	–	M*	Carbide TiAlN	M*	Carbide TiAlN	K*	Carbide TiAlN	G*	Carbide TiAlN
S	**High Temp Alloy**	–	G*	Carbide TiAlN	G*	Carbide TiAlN	K*	Carbide TiAlN	G*	Carbide TiAlN
S	**Titanium Alloys**	–	T	Carbide TiAlN	T	Carbide TiAlN	T	Carbide TiAlN	T	Carbide TiAlN
M	**Austenitic Stainless Steel** 304, 316, etc.	–	E	Carbide Alcrona	E	Carbide Alcrona	K	Carbide Alcrona	G*	Carbide Alcrona
M	**Ferritic Martensitic Stainless Steel** 416, 420, 17-4PH, 15-5PH, etc.	–	N or E	Cermet or Carbide Alcrona	E	Cermet or Carbide Alcrona	K	Cermet or Carbide Alcrona	G	Cermet or Carbide Alcrona
K	**Ductile Cast Iron** Spheroidal - GS500	Below 130	V	Carbide Alcrona	V	Carbide Alcrona	K	Carbide Alcrona	V	Carbide Alcrona
		130 Above		Cermet Alcrona		Cermet Alcrona		Cermet Alcrona		Cermet Alcrona
K	**Grey Cast Iron** GC15 - GC20 - GC25 - GC35	–	V	Carbide TiAlN	V	Carbide TiAlN	K	Carbide TiAlN	V	Carbide TiAlN
N	**Bronze Brass Copper**	Below 300	E	Carbide Uncoated	E	Carbide Uncoated	K	Carbide Uncoated	G	Carbide Uncoated
N	**Aluminum**	Below 7% Si	V	Carbide Uncoated	V	Carbide Uncoated	V	Carbide Uncoated	G	Carbide Uncoated
		Above 7% Si	G	PCD Uncoated	G	PCD Uncoated	G	PCD Uncoated		PCD Uncoated

*Contact our Application Engineering department for special geometries to improve tool life.

Lead-in Angle Information

Straight Flute

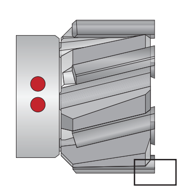
Lead-in	Angles	Chip Evacuation	Description
A	45°, 8°	↔	Lead-in can be used to improve finish.
F	90°	↔	Can be used for stock removal at the bottom of the hole. Reduce the feed by 40% of the values on the recommended cutting data pages.
G	45°	↔	Standard and suitable for most materials.
L	75°	↔	May provide improved straightness. Reduce the feed by 40% of the values on the recommended cutting data pages.
N	20°	↔	Ideal for through holes. It is possible to increase the feed up to 100% of the values on the recommended cutting data pages.
T	30°	↔	Suitable for titanium based alloys.
V	45°	↔	Suitable for most materials and increases tool life
K	80°, 0.10x45°	↔	Excellent at breaking small chips that are easy to evacuate in blind hole applications. Requires 50% increased feed rate, which will result in reduced tool life when compared to other leads.



Straight

Helical Flute (Left-Hand) - *Through Hole Applications Only*

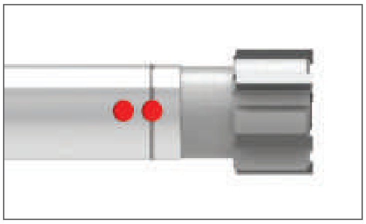
Lead-in	Angles	Chip Evacuation	Description
E	25°	→	Standard and suitable for most materials. **NOTE:** Through hole applications only.
M	45°	→	May provide better penetration rates in steels over 200 BHN. **NOTE:** Through hole applications only.



Helical

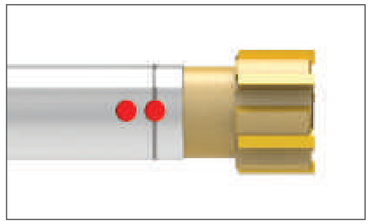
Coatings, Cutting Materials, and Dimple Indicators

Coating Information



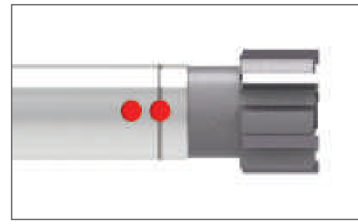
Uncoated

Ideal for non-ferrous applications



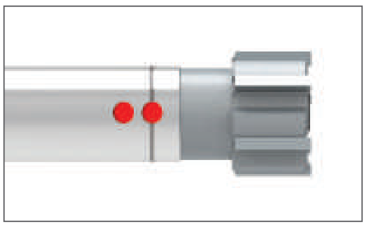
TiN (N)

Ideal for general purpose applications



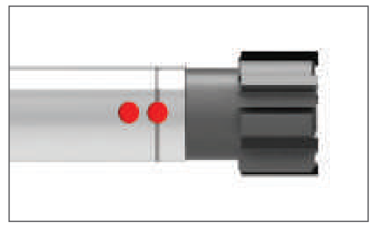
TiAlN (A)

Provides higher heat resistance to improve tool life



TiCN (C)

Provides improved surface finish



Alcrona (K)

Provides excellent wear resistance and can help increase cutting speeds

Cutting Material Information

Material	Indicator	Details
Carbide	K	A fine-grain carbide suitable for all conventional reaming applications. Recommended where rigidity is not excellent and speeds must be reduced.
Cermet	S	Cermet provides high wear resistance and is recommended for abrasive and increased speed applications. Not recommended for poor rigidity or interrupted cuts.

Dimple Indicators

Material	Indicator	Replaceable Head Style	Monobloc Style	Cutting Ring Style
Carbide	**Two Dimples**			
Cermet	**Two Dimples with Line**			

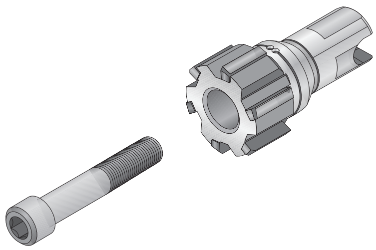
NOTE: The dimple location also indicates which 2 cutting teeth are 180° opposed

Replaceable Head Reamers

Product Overview

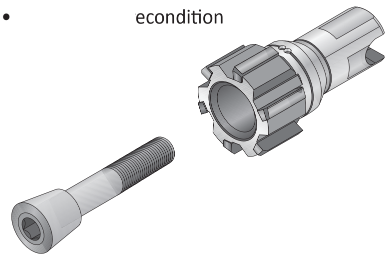
Fixed Heads

- Non-expanding diameter
- Locking screw is straight (no taper)
- Allows for on-machine replacement
- Capable of H7 tolerance on diameter
- Available in straight and left hand helical flutes
- Available for recondition



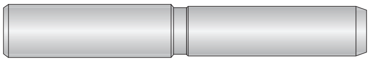
Expandable Heads

- Expandable diameter (1% of nominal diameter) to accommodate for wear
- Conical locking screw
- Requires setup for diameter
- Capable of tight diameter tolerance (± 0.0002" (0.005 mm))
- Available in straight and left-hand helical flutes
- econdition



Mandrels

- Available in short, standard, and long lengths
- Reamer head design allows multiple diameters to be used within the same mandrel, ***which reduces inventory requirements***
- The same mandrel can use both fixed and expandable heads
- Coolant options are offered for both through and blind hole scenarios




Uncoated

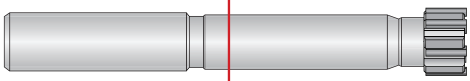

TiN Coated

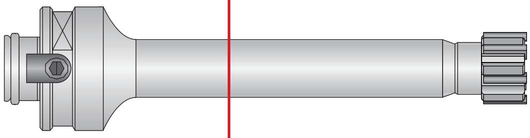

TiAlN Coated


TiCN Coated

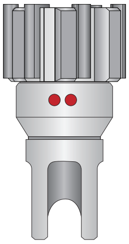

Alcrona Coated

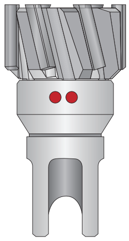
Mandrel Shanks Available:
- Cylindrical
- Modular Connection


Short Length


Standard Length


Long Length


Straight Flute

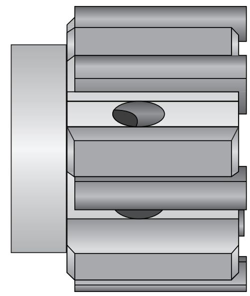

Left-Hand Helical Flute

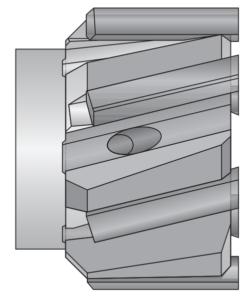
Type of Head	Coated/ Uncoated	Lead Time in Work Days (based on number of pieces)		
		Up to 5	6 - 19	20+
Fixed	Coated	15	25	25
	Uncoated	10	20	20
Expandable	Coated	20	25	30
	Uncoated	15	20	25

Monobloc Style Reamers

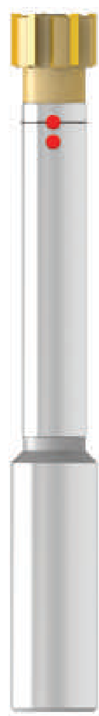
Product Overview

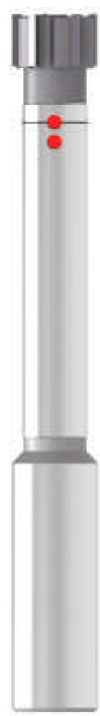
Monobloc Reamer Features
• Diameter range: 0.2283" - 1.2638" (5.80 mm - 32.10 mm) • Available with straight or left-hand helical flutes • Expandable up to 1% of nominal diameter • Available with cylindrical shanks only • Work day lead time 20 - 25 days • Available for recondition

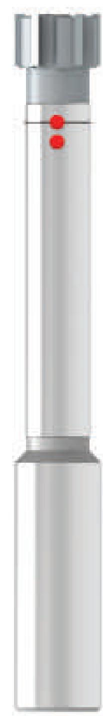

Straight Flute

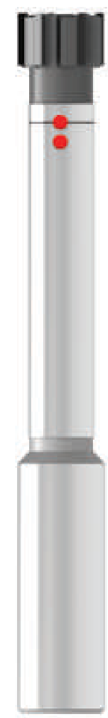

Left-Hand Helical Flute


Uncoated

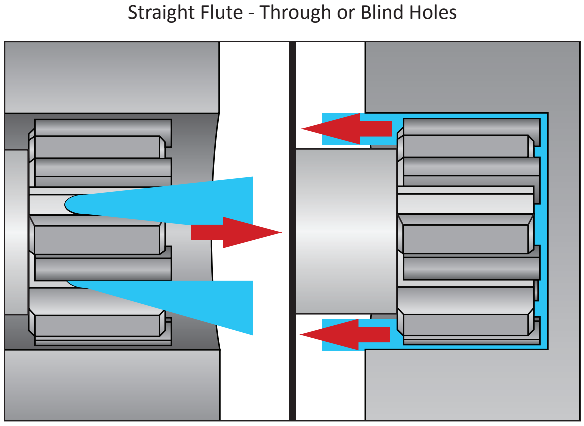

TiN Coated


TiAlN Coated


TiCN Coated

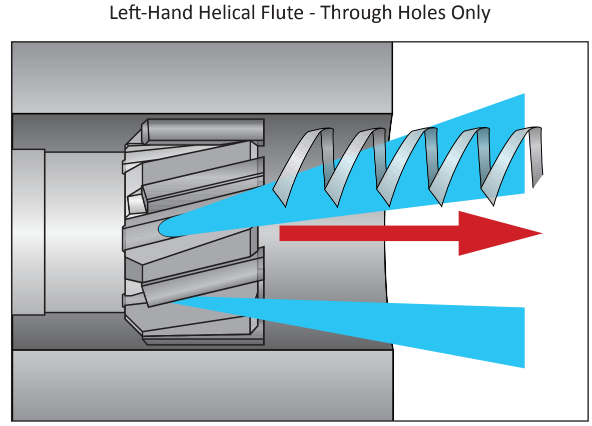

Alcrona Coated

Straight Flute - Through or Blind Holes



Use for either through hole or blind hole applications. The coolant flow determines the direction of the chip evacuation.

Left-Hand Helical Flute - Through Holes Only

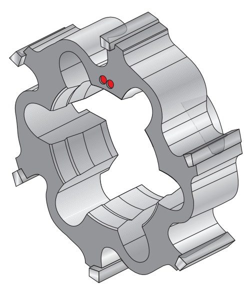


Use when reaming through hole applications. The cutting action of the helical flutes forces the chips forward for evacuation.

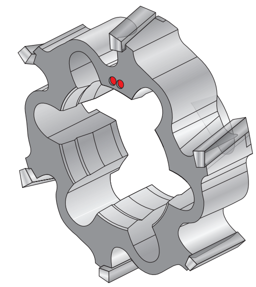
Cutting Ring Style Reamers

Product Overview

Cutting Ring Reamer Features
• Diameter range: 0.6929” - 7.8972” (17.60 mm - 200.59 mm) • Available with straight or left-hand helical flutes • Expandable up to 4% of nominal diameter • Mandrels are available for both through holes or blind holes • Work day lead time 20 - 25 days • Available for recondition

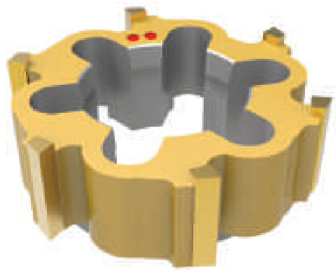


Straight Flute



Left-Hand Helical Flute

Uncoated

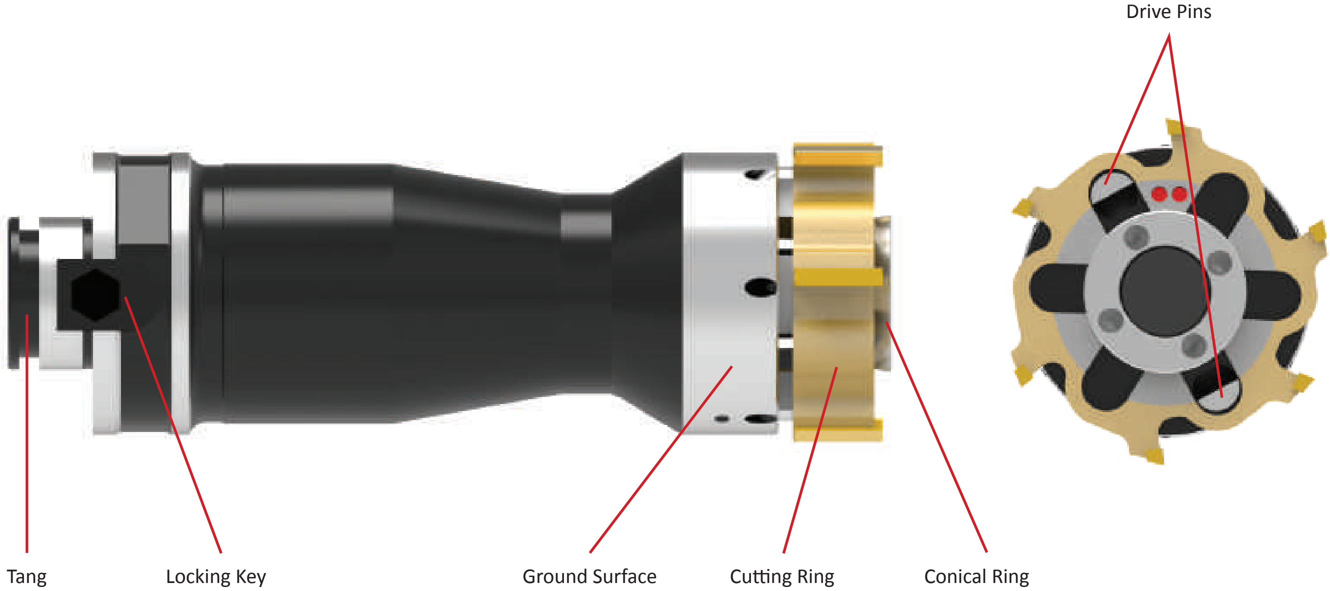


TiN Coated

TiAlN Coated

TiCN Coated

Alcrona Coated

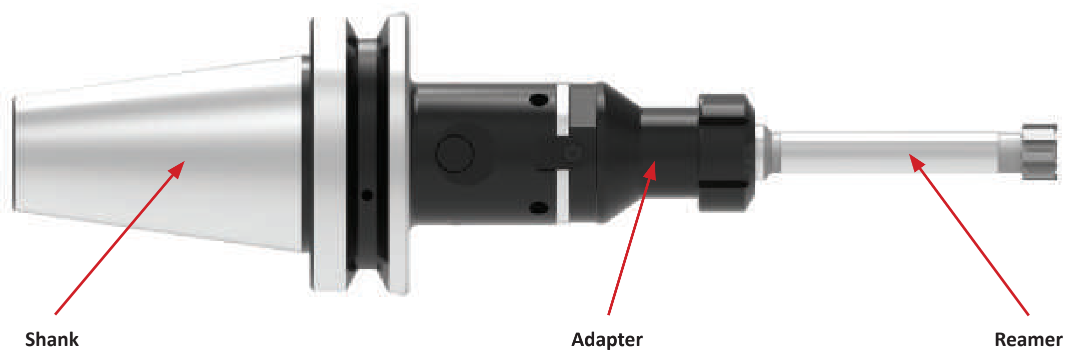


Radial Adjusting Shanks

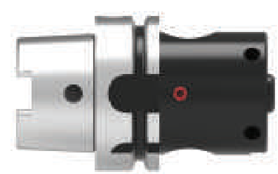
Large range of shanks for different machine types	Highly adjustable for improved concentricity	All shanks are available with through coolant

All the Pieces You Need

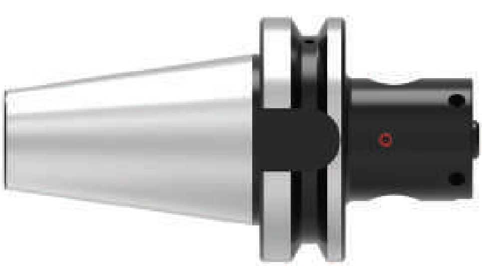
Modular System courtesy of cerit



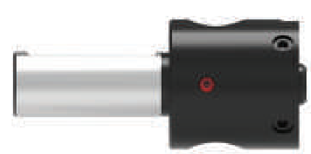
DIN 69871/1 B and A



HSK-A DIN 69893/1

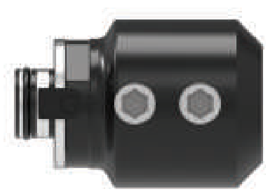


JMTBA MAS-403 BT B and BT



Straight

Collet Chuck Adapter



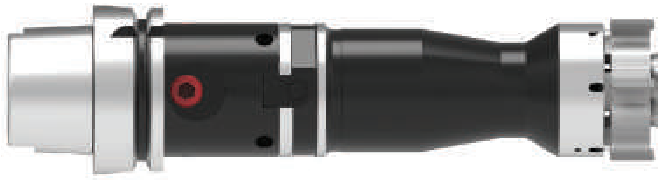
Cylindrical Shank Adapter

Radial Adjusting Shanks

Setup Information

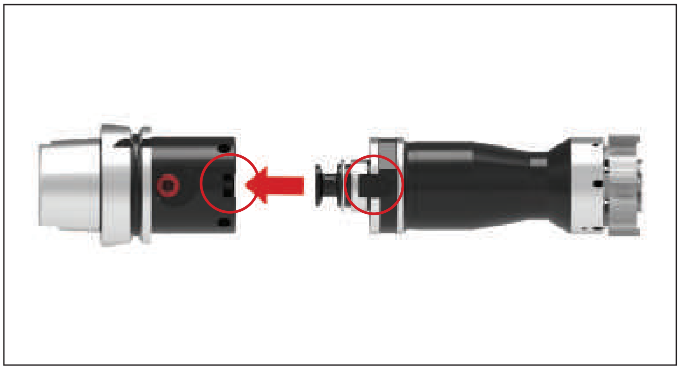
Radial Adjusting Shanks and Ring Style Arbors

The following is a quick guide for setting up a radial adjusting shank and a ring style reamer. The ring reamer arbor does not contain the tang needed to connect to the shank. The tang must first be removed from the shank and then installed into the reamer arbor (demonstrated below).



Step 1:
The tang comes installed with the shank. Loosen the clamping screw on each side and remove the tang from the shank.

Step 2:
Thread the tang into the back end of the ring arbor. Use a bench vise and wrench to tighten.



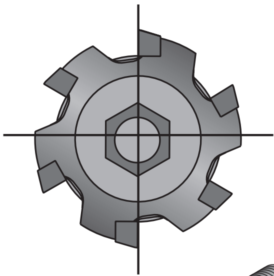
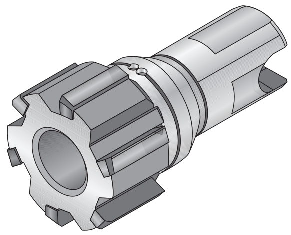
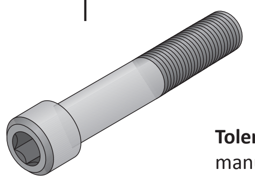
Step 3:
Assemble the ring arbor to the shank. With the clamping screws still loosened, align the key on the arbor to the keyway on the shank.

Step 4:
Once the ring arbor is connected with the shank, tighten the clamping screws to secure the tang back into place.

Setup Information

Replaceable Head Style

Fixed Head Style

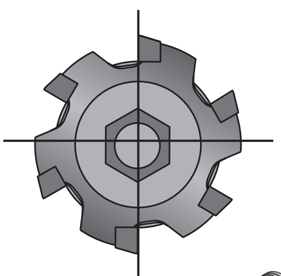
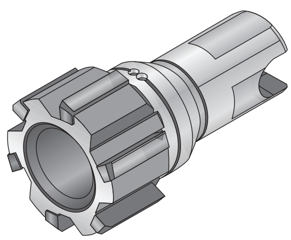
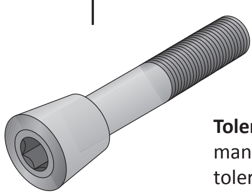




Tolerance: Fixed reamers are manufactured at 2/3 of the tolerance

Recommended Tightening Torque for Fixed Head Reamer (7400 / 7700)

Imperial		Metric	
D_1 Range (inch)	Torque (in-lbs)	D_1 Range (mm)	Torque (N-m)
0.465 - 0.575	22.1	11.80 - 14.60	2.5
0.575 - 0.693	33.6	14.61 - 17.60	3.5
0.693 - 0.850	44.3	17.61 - 21.60	5.0
0.851 - 1.047	62.0	21.61 - 26.60	7.0
1.048 - 1.283	88.5	26.61 - 32.60	10.0
1.284 - 1.598	106.2	32.61 - 40.60	12.0
1.599 - 1.992	141.6	40.61 - 50.60	16.0
1.993 - 2.386	177.0	50.61 - 60.60	20.0

Expandable Head Style

Tolerance: Expandable reamers are manufactured to the middle of the tolerance

Expanding Heads Adjustment

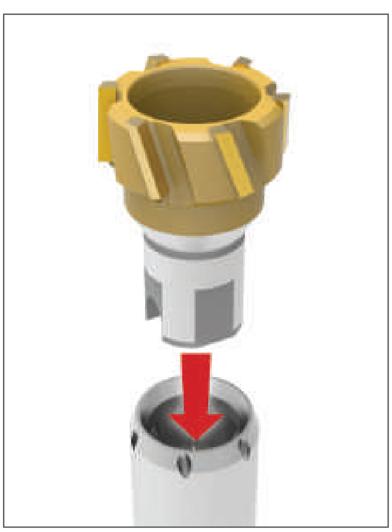
When the size reaches its lower tolerance, the head can be adjusted to compensate for wear to the cutting edges. This operation can be repeated several times until the surface finish of the hole deteriorates to an unacceptable level.

Adjustment Procedure

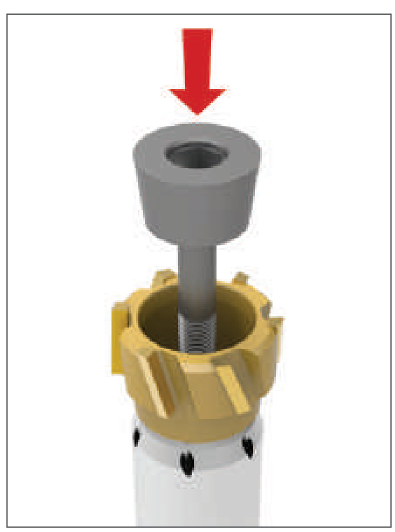
Slowly turn the right-hand threaded screw clockwise while checking the diameter setting of the reamer with a micrometer. When the required diameter is achieved, the tool is ready for use.

Replaceable Head Reamer Assembly

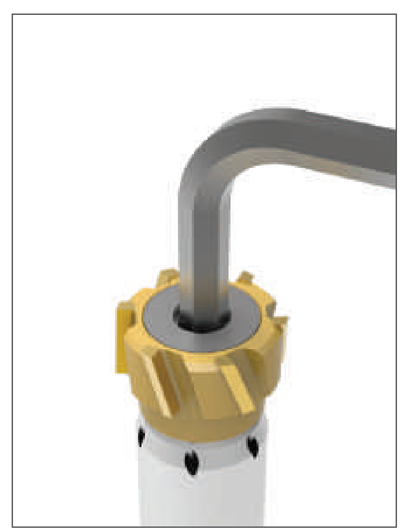
Fixed and Expandable Styles



Step 1:
Insert the replaceable reamer head into the mandrel.



Step 2:
Insert the screw into the reamer head opening to secure it to the mandrel.

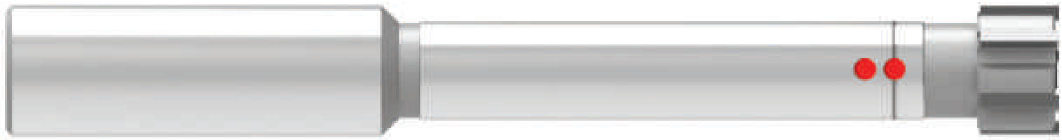


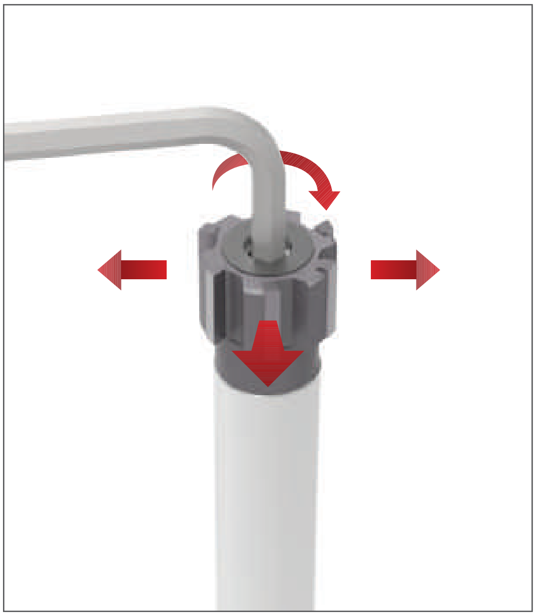
Step 3:
Tighten the screw.

NOTE: We recommend lubricating the thread and the conical surface of contact between the reamer head and the screw with antifriction Molycote® grease.

Setup Information

Monobloc Style





Tolerance

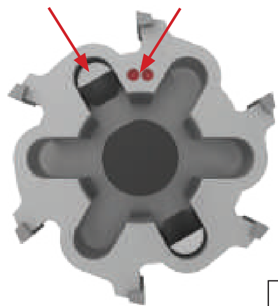
All monobloc reamers are ground to the requested diameter and set in the middle of the hole tolerance ready for use.

Adjustment

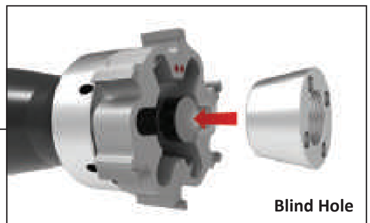
The adjustment must be made to compensate for wear to the cutting edges when the size reaches its lower tolerance. This operation can be repeated several times until the surface finish of the hole deteriorates to an unacceptable level. Then the reamer must be reground. The maximum expansion is about 1% of the diameter.

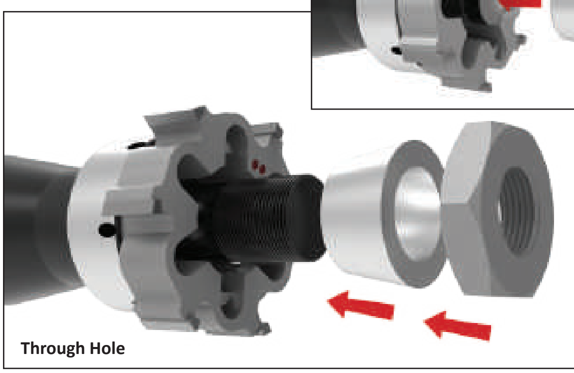
Setup Information

Cutting Ring Style



Step 1:
With the drive pins assembled, insert the cutting ring onto the mandrel. Make sure the dimples are at the 12:00 position with the drive pin at the 11:00 position.





Step 2:

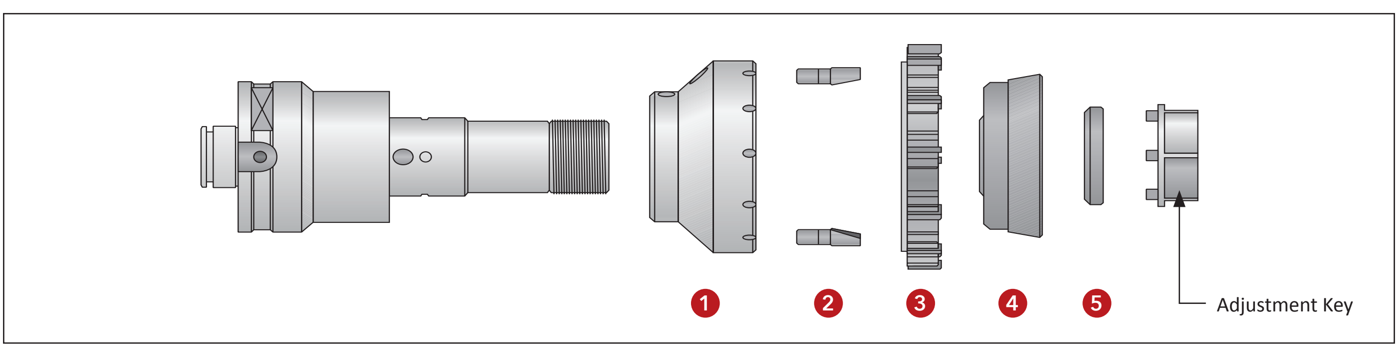
Insert the conical ring. Tighten the lock nut to set the desired reamer size (left-hand thread). Then loosen the lock nut slightly until it “clicks” against the drive wall.

NOTE: We recommend lubricating the thread and the conical surface of contact between the cutting ring and the conical ring with antifriction Molycote® grease.

Adjustment Procedure

1. Turn the conical ring slowly using an adjustment key (left-hand thread). Adjustment keys are supplied with reamers from diameter 17.60 mm to 40.59 mm.
2. Check the diameter setting of the cutting ring with a micrometer.
3. When the required diameter is achieved, unscrew the conical ring until there is a click and the drive pins are in traction in the opposite direction to the cutting action of the reamer. The reamer is ready for use.

For Diameter Range: 100.60 mm - 200.59 mm



Assembly

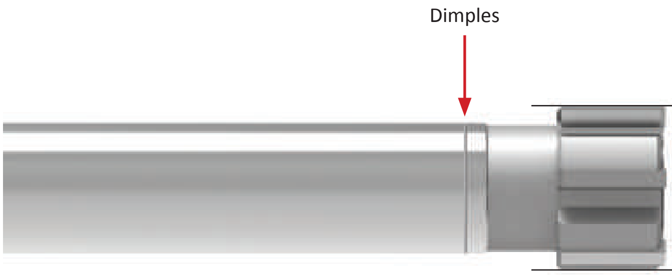
1. With the drive pins (2) assembled, mount the flange (1) onto the mandrel. Assemble the cutting ring (3) so the slot on the left side of the dimple is mounted onto the drive pins (2). Insert the conical ring (4).
2. Screw the ring nut (5) onto the mandrel and tighten manually so the conical ring (4) makes contact with the cutting ring (3). The thread is left handed.

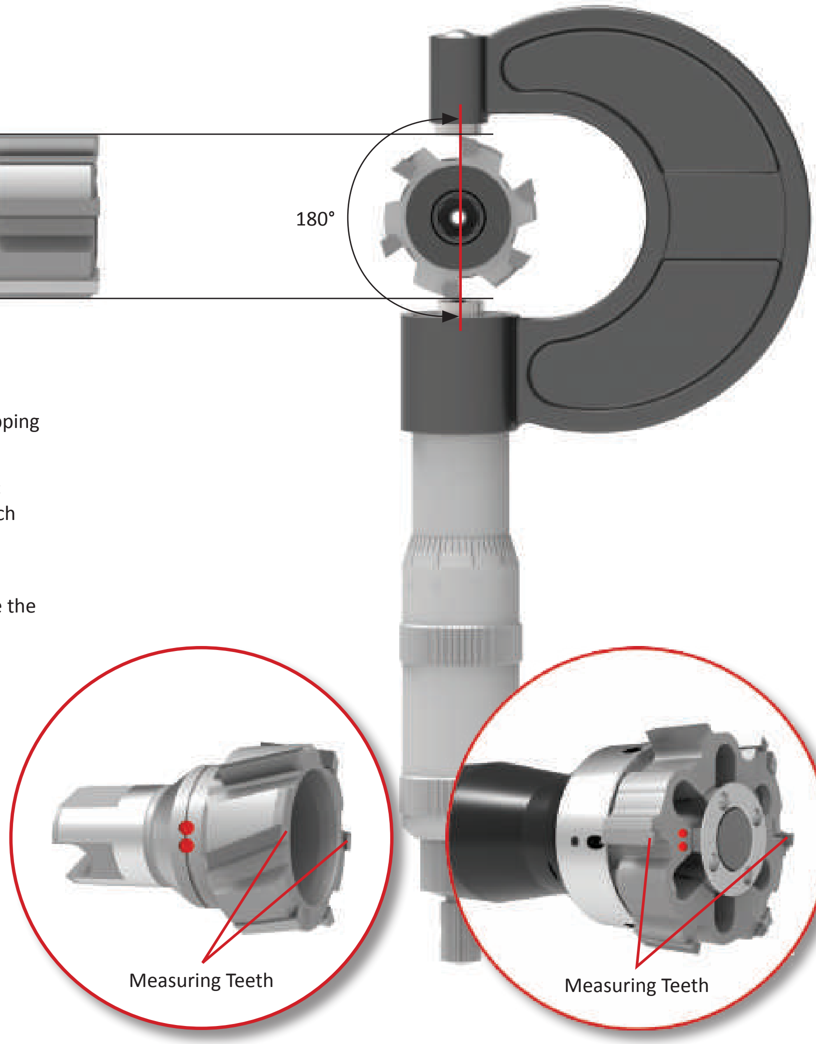
   NOTE: We recommend lubricating the thread and the conical surface of contact between the cutting ring and the conical ring with antifriction Molycote grease.

Adjustment Procedure

1. Turn the ring nut (5) slowly using a pin spanner.
2. Check the diameter setting of the cutting ring with a micrometer. Make sure the drive pins (2) are in traction and in the opposite direction of the cutting action of the reamer.
3. When the required diameter is achieved, the tool is ready to use.

Diameter Measurement



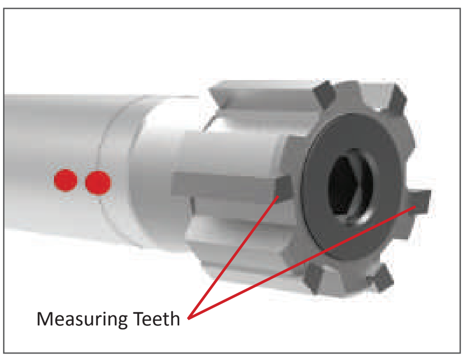


Using the Measuring Teeth

With the reamer assembled, use a presetter or micrometers to measure the reamer diameter using the opposing 180° teeth. A presetter (with at least 2 µm resolution) is preferred to avoid chipping the cutting edges.

NOTE: Only two cutting teeth are 180° opposed. The asymmetric spacing of the other cutting teeth will not induce harmonics, which prevents the tool from creating chatter.

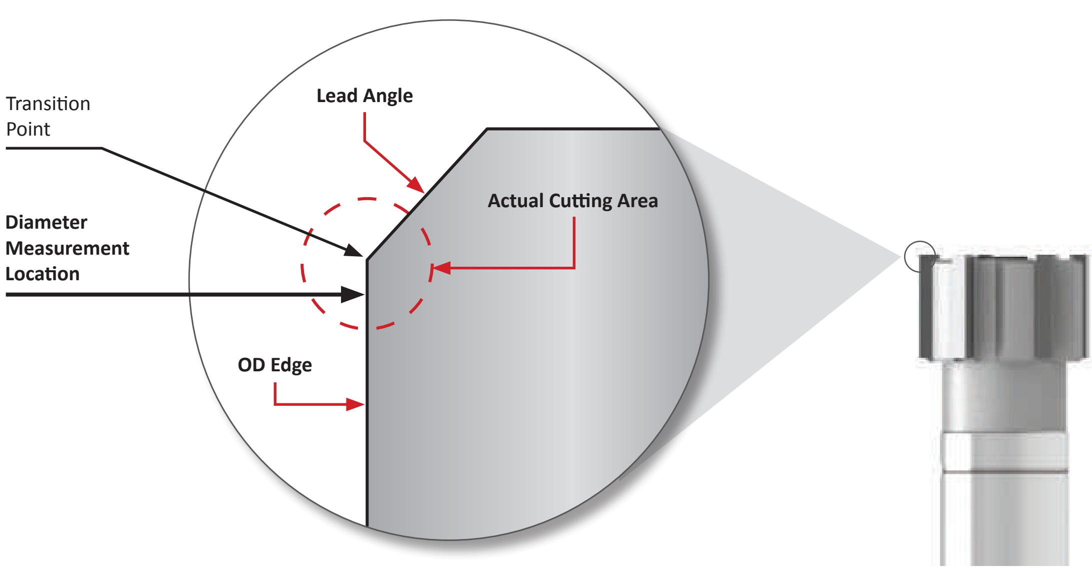
The red dimples indicate which two opposing teeth are the measuring teeth. All S.C.A.M.I. Reamers have a dimple to indicate the 180° opposing teeth.



Where to Take the Measurement

When measuring the diameter, take the measurement from the area of the cutting tooth just below the transition from the lead angle to the OD edge. See the illustration below.

The back side of the OD edge has a back taper. This is why measuring from the location just below the lead angle/OD edge transition point results in the most accurate measurement (before the taper begins).



TIR Measurement

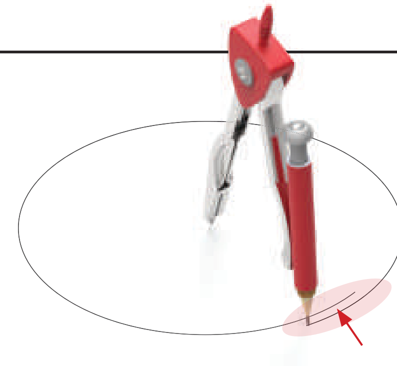
What is TIR?

Total indicator runout (TIR) refers to the distance to which the reamer is cutting off-center. In an ideal situation, the tool would begin in the exact center of the hole, and it would then rotate and cut in a perfect circle. This would result in a TIR of 0.

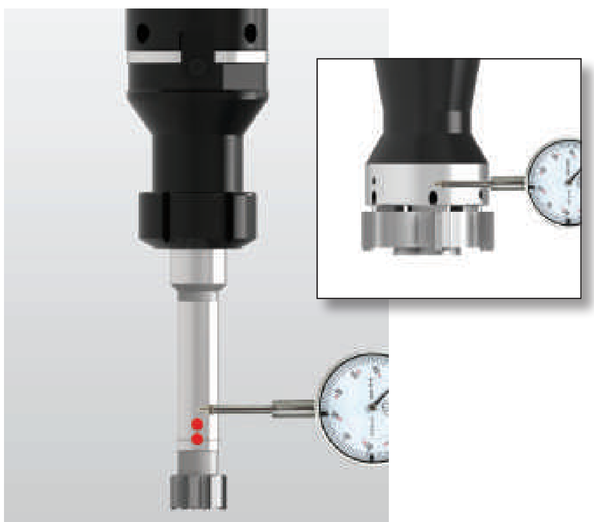
Because a perfect TIR of 0 is not practical, the goal is to maintain a TIR as close to 0 as possible. The closer the TIR is to 0, the better the reamer will perform.

Allied Machine recommends a TIR of < 0.0005" (0.013 mm).

Think of attempting to draw a perfect circle with a drafting compass, but the pencil runs slightly outside the point where the circle began because the center point shifted during the pencil's path. This slight area of overlap would be the TIR.



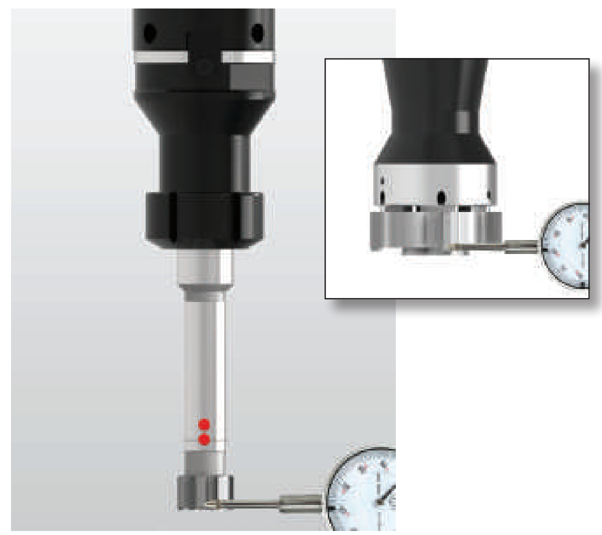
TIR: How far from center the tool will move during its path



Step 1:

Check the TIR first on the mandrel (or ground) area of the reamer. Center the indicator in line with the dimple.

Measure the TIR by rotating the tool until the indicator reaches the highest value.

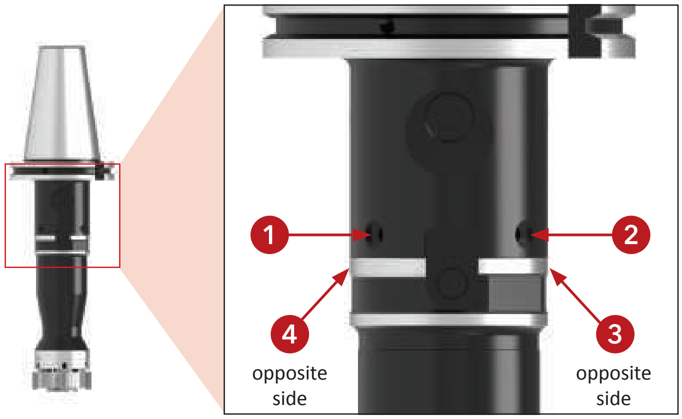


Step 2:

Next, check the TIR on the cutting teeth of the reamer.

NOTE: Rotate the tool counterclockwise to avoid chipping the cutting teeth with the indicator.

TIR Adjustment



Step 1:

Place the tool into the machine spindle. Make contact with the four radial adjustment screws in a concentric fashion (this results in equal pressure surrounding the tool).

Tighten #1, then #3, followed by #2 and #4.

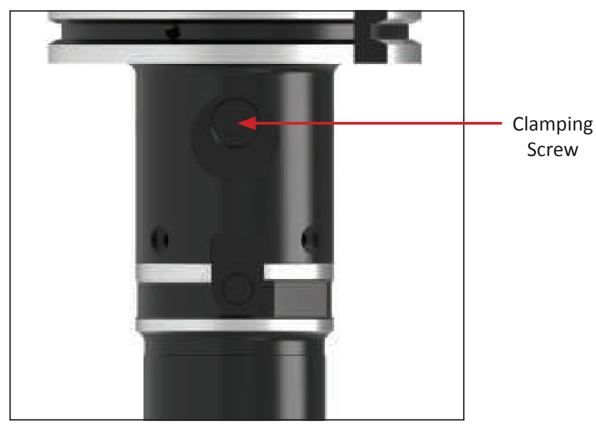
Step 2:

Swipe the dial indicator around the ground portion of the arbor near the coolant outlet holes to verify the TIR.

The TIR should be within 0.0005” (as close to 0 as possible). This will ensure the TIR check on the cutting teeth will be more true. It also means the arbor is running true to the shank.

Step 3:

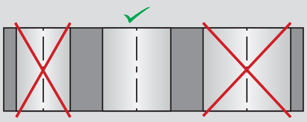
Once the TIR is checked on the arbor, check the TIR on the cutting teeth. Rotate the tool counterclockwise to avoid chipping the cutting teeth.



Step 4:

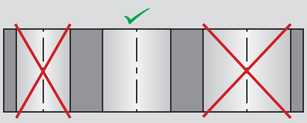
Tighten down the central clamping screws. During the tightening, the tool body will shift slightly. Repeat the TIR check on the cutting teeth and adjust as necessary.

Troubleshooting Guide



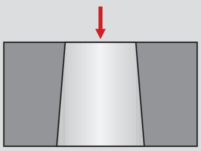
Oversized Hole

- Reamer is running eccentric to the center of the machine spindle ▶ Use modular system with radial adjustment
- Excessive misalignment causing reamer to cut on back taper ▶ Fix the misalignment
- Material buildup on cutting edges ▶ Replace the coolant or change the cutting speed
- Reamer diameter is too large ▶ Use smaller reamer or regrind existing reamer



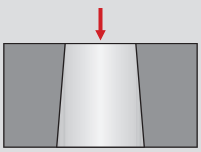
Undersized Hole

- The reamer diameter is too small ▶ Use larger reamer
- The reamer diameter is worn ▶ Expand, regrind, or replace the reamer
- The coolant is not suitable ▶ Replace the coolant
- Stock allowance is too small ▶ Increase the stock allowance
- The cutting speed is too low ▶ Increase the cutting speed



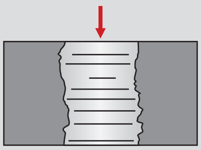
Tapered Hole

- Excessive misalignment ▶ Correct the misalignment



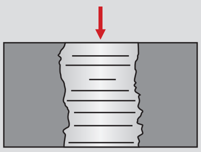
Burr at Hole Entry

- Excessive misalignment ▶ Correct the misalignment



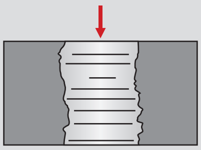
Hole is Not Straight

- Concentricity and alignment error between the workpiece and the tool ▶ Correct the misalignment and use the modular system with radial adjustment
- Asymmetrical cutting or angled surfaces ▶ Create a chamfer on the lead-in



Poor Hole Finish

- One cutting edge is chipped ▶ Regrind the reamer
- The lead-in is irregular ▶ Regrind the reamer
- Back taper on the cutting edge is too great ▶ Regrind the reamer
- Excessive misalignment ▶ Correct the misalignment or use the modular system
- Cutting data is not correct ▶ Verify the cutting data
- Poor chip evacuation ▶ Verify the coolant volume and pressure or use through-tool coolant



Reamer Creates Excessive Torque Loading

- Back taper on the cutting edge is too small ▶ Regrind the reamer
- The radially ground land is too wide ▶ Regrind the reamer
- The coolant is not suitable ▶ Replace the coolant

SECTION

D

Roller Burnishing

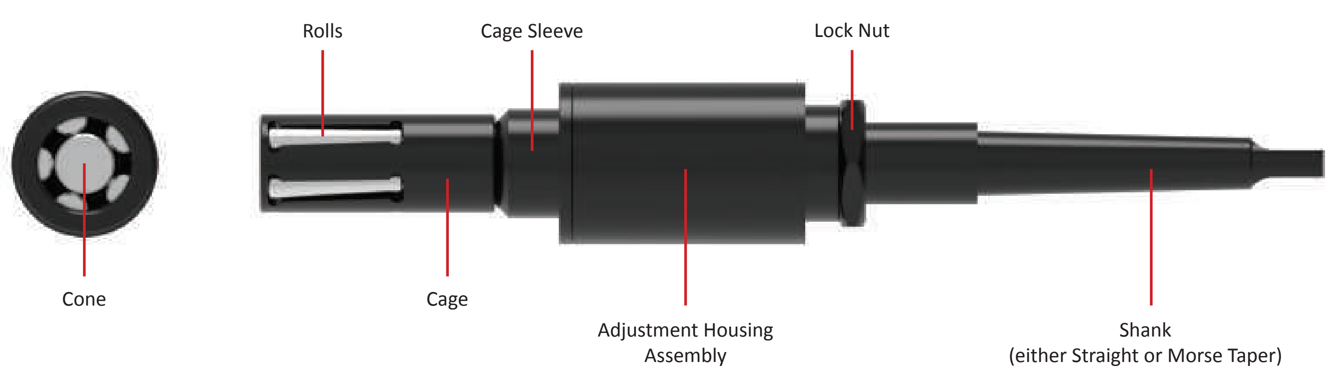
Product Offering

Through Hole Style
0.1555" - 6.5315" (3.95 mm - 165.90 mm)

Blind Hole Style
0.2319" - 6.5315" (5.89 mm - 165.90 mm)

Advantages of the Roller Burnishing Tool

- ***Provides accurate size control***
  tolerances within 0.0005" or better (depending on variables such as material)
- ***Produces fine surface finishes***
  between 1 - 10 microinches Ra
- ***Increases surface hardness***
  by 5 - 10% or more
- ***Performs a much cleaner operation***
  than honing or other abrasive finishing methods
- ***Provides versatility***
  because the operation can be performed on any rotating spindle
- ***Eliminates the need for slower and costly finishing processes and secondary operations***
  such as grinding, honing, lapping, etc.

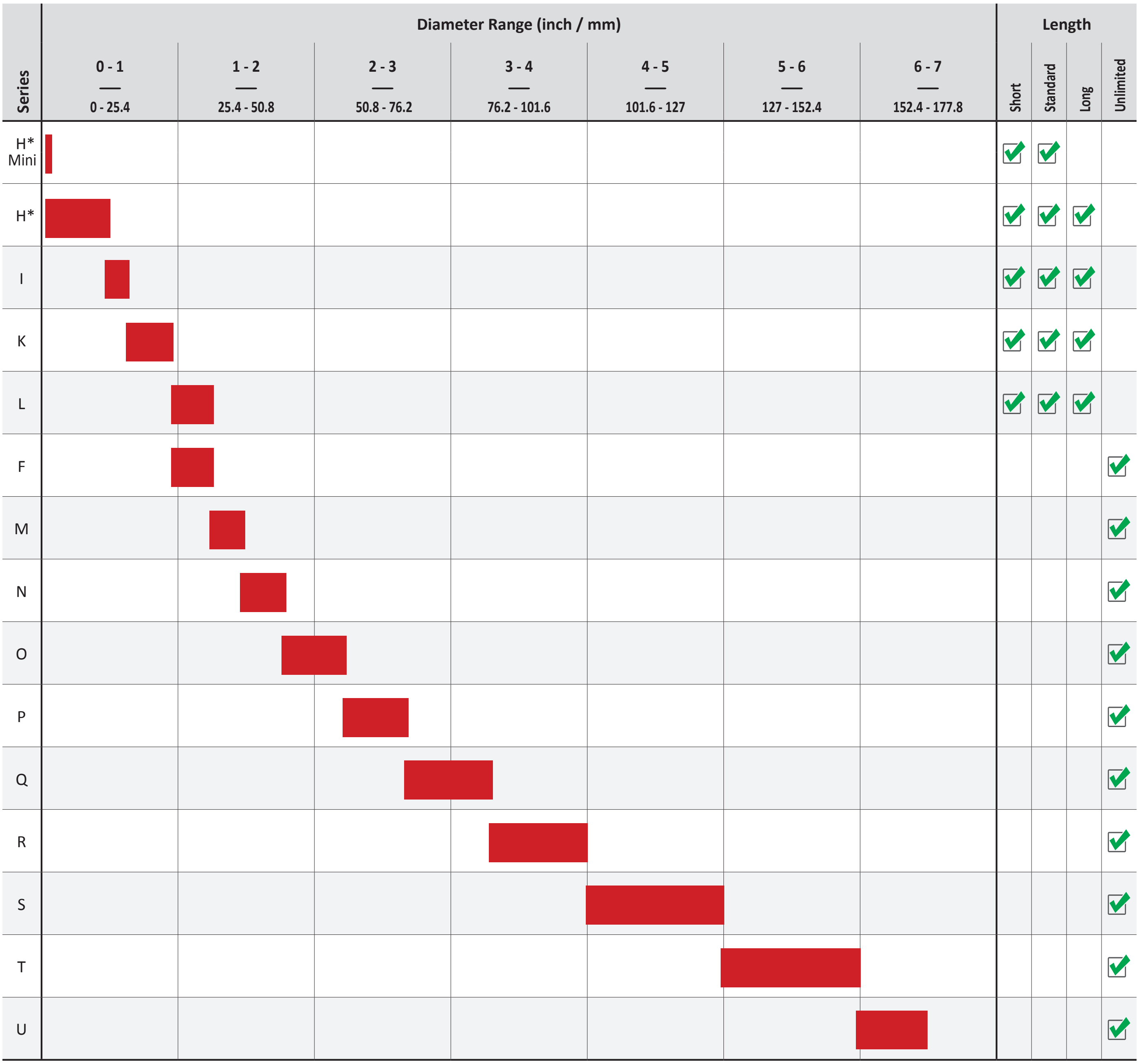


The Tool Components

All roller burnishing tools (both through hole and blind hole) are composed of the basic burnisher assembly including:

- Cage
- Cone
- Rolls
- Shank (either straight or Morse Taper)

Product Selection Guide



*For H series: Through hole tools start at 0.1555" (3.95 mm) and blind hole tools start at 0.2319" (5.89mm)

When to ORDER UP

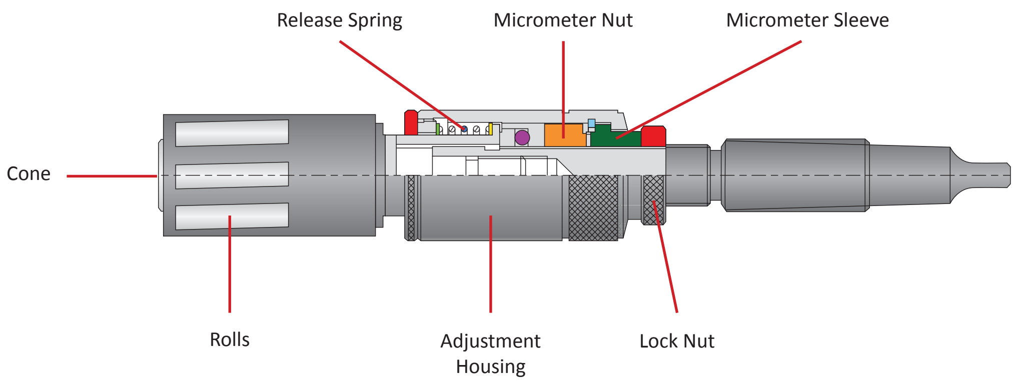
In some cases, there will be a diameter overlap between a series and the series after it. If the diameter you need falls into this overlap, choose the higher of the two series.

Example:
You need a 24.64mm diameter tool. This diameter falls into both the K series and the L series.

- K series diameter range = 16.60 mm - 24.74 mm
- L series diameter range = 24.54 mm - 31.16 mm

In this scenario, you would choose the L series tool that covers the 24.64 diameter.

Diameter Adjustment



Adjustment

The roller burnishing tool incorporates a shank, a body, and a planetary system of conical rolls that are evenly spaced by a retaining cage.

1. Unscrew the lock nut.
2. Pull the housing toward the lock nut and rotate to increase or decrease the diameter.
3. Tighten the lock nut.

IMPORTANT: As you increase the diameter, the cone moves forward, pushing the rolls outward. Because of this, the cone will protrude from the end of the cage, decreasing the clearance available in blind holes (see Figures 1 and 2).

Refer to chart below for clearance values.

Figure 1

Through Hole Tool **Blind Hole Tool**

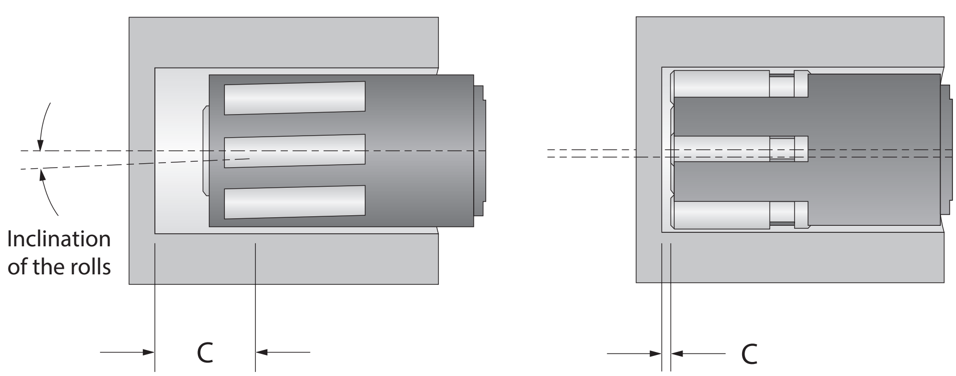


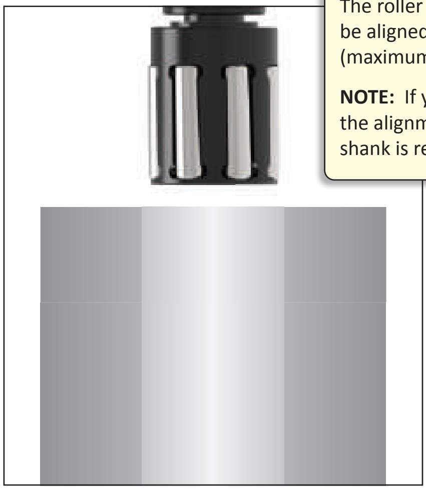
Figure 2

Larger Diameter = Extruded Cone = Decreased Clearance

Adjustment Range		Clearance (C)		
		Through Holes		Blind Holes
Imperial (inch)	Metric (mm)	Rolls 701	Rolls 704 / 707	Rolls 708
0.1850 - 0.2315	4.70 - 5.88	–	2.40	–
0.2319 - 0.3728	5.89 - 9.47	–	2.40	0.60
0.3732 - 0.6236	9.48 - 15.84	–	2.40	0.60
0.6240 - 1.1236	15.85 - 28.54	5.40	3.20	1.00
1.1240 - 1.8385	28.55 - 46.70	9.50	3.20	1.00
1.8390 - 3.3386	46.71 - 84.80	9.50	4.00	1.00
3.3390 - 6.5315	84.81 - 165.90	10.30	4.70	1.00

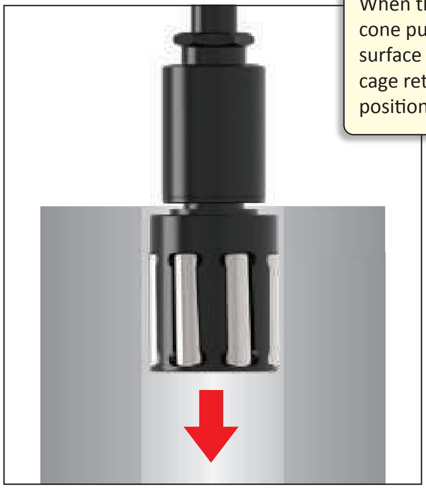
How it Works

Roller Burnishing Tools

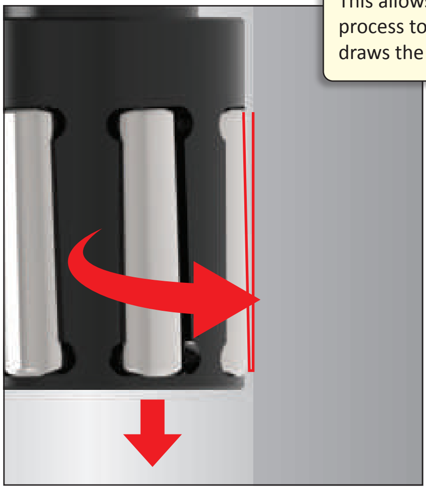


The roller burnishing tool must be aligned with the workpiece (maximum error = 0.1 mm).

NOTE: If you are uncertain about the alignment error, a floating shank is recommended.



When the tool is in the hole, the cone pushes the rolls onto the surface to be burnished. The cage retains the rolls in an angled position to the axis of the tool.

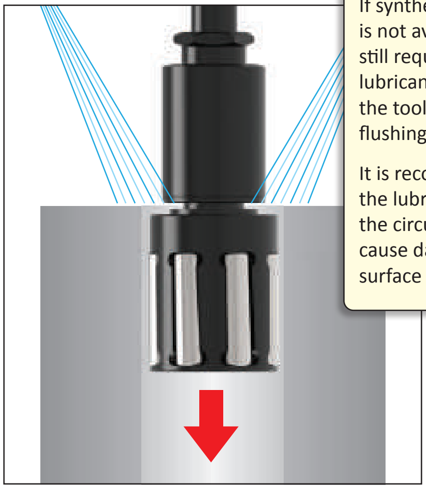


This allows the pressure of the process to generate a self-feed that draws the tool into the hole.

NOTE: When self-feed is not required (blind hole applications), you must use a cage that holds the rolls parallel with the axis of the tool.

If the machine feed is stopped or slower than the natural rate, the rolls pull the cage (held in position by the release spring) and then release from the surface of the hole.

When retracting the tool, the release spring ensures the cage is withdrawn from the cone, which removes pressure from the rolls. This allows the tool to be retracted at a high feed rate.



If synthetic or water soluble coolant is not available, a lubrication is still required. A plentiful supply of lubricant must be fed directly to the tool in order to provide a good flushing and cleaning action.

It is recommended to fit a filter to the lubrication system to prevent the circulation of chips, which may cause damage to the burnished surface and/or the tool.

Notes

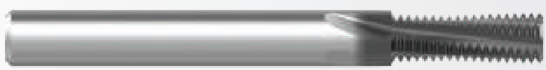
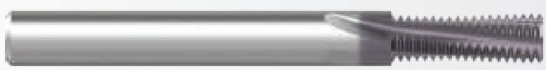
SECTION

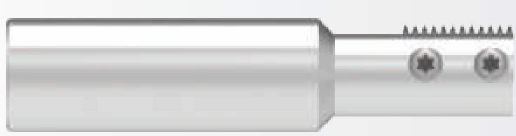
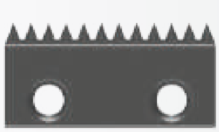
E

Threading

High Performance Threading Solutions

THREAD MILLING DONE *RIGHT*

Solid Carbide Thread Mills		Notes
AccuThread® 856		• Allied Machine's proprietary AM210® coating yields a 25-50% increase in tool life over competitor products • Standard cutting lengths allow for multiple applications without the need for special thread mills • Helical flute offers increased strength and rigidity when cutting forces are applied
ThreadMills USA™		• Helical flute offers increased strength and rigidity when cutting forces are applied • High quality for consistent, predictable production • Coolant-through options available • TiAlN coating improves tool life versus uncoated tools
AccuThread® T3		• Allied Machine's proprietary AM210® coating yields a 25-50% increase in tool life over competitor products • Standard cutting lengths allow for multiple applications without the need for special thread mills • Helical flute offers increased strength and rigidity when cutting forces are applied

Indexable Insert Thread Mills			Notes
AccuThread® 856 Bolt-in Style			• Thread mill holders are manufactured from stainless steel that is engineered to dampen vibration during operation • Extensive range of thread forms with two thread lengths • Can produce left-or right-handed threads
AccuThread® 856 Pin Style			• Patented pin style locking system ensures unsurpassed repeatability • Thread mill holders are manufactured from stainless steel that is engineered to dampen vibration during operation • Extensive range of thread forms with two thread lengths
AccuThread® 856 Indexable Inserts	 Bolt-in Style	 Pin Style	• Full profiles present on all inserts allow 100% thread form against 65-75% for tapping • Allied Machine's premium carbide allows for extended tool life while providing high-quality thread forms • Allied Machine's proprietary AM210® coating yields a 25-50% increase in tool life over competitor products

Port and Thread Finishing Kits

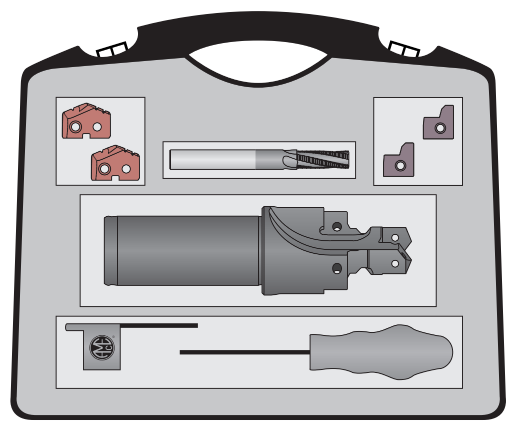
WE HAVE A KIT FOR THAT

Kits aren't for everyone, but if you work on different projects from day to day, you need to ***be prepared for the work tomorrow will bring.***

The Complete Package

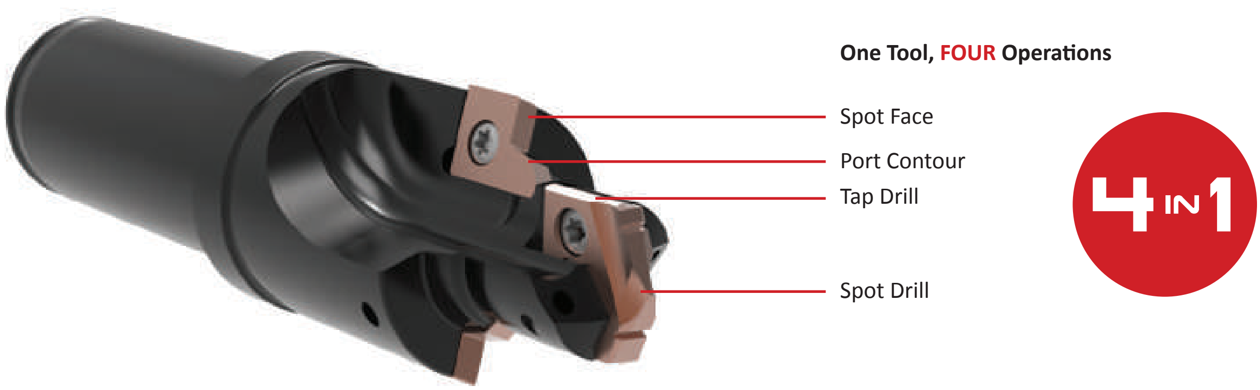
Producing fully finished threaded hydraulic ports has never been easier. The Port and Thread Finishing Kit includes the AccuPort 432® port contour cutter with a dedicated AccuThread® 856 solid carbide thread mill in a single kit. You also receive the T-A® inserts and port form inserts needed to complete the assembly.

Port kits incorporate the AccuThread 856 solid carbide thread mills to increase the manufacturing flexibility by allowing hydraulic ports to be produced in just two operations. In addition, where a unique port profile is required, Allied Machine provides a dedicated special tooling solution using our extensive tool design and manufacturing experience to meet precise specifications.



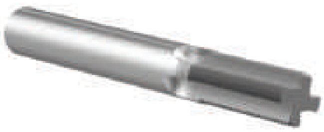
NOTE: See Section A92 of our product catalog for the complete list of Port and Thread Finishing Kits.

One Tool, FOUR Operations

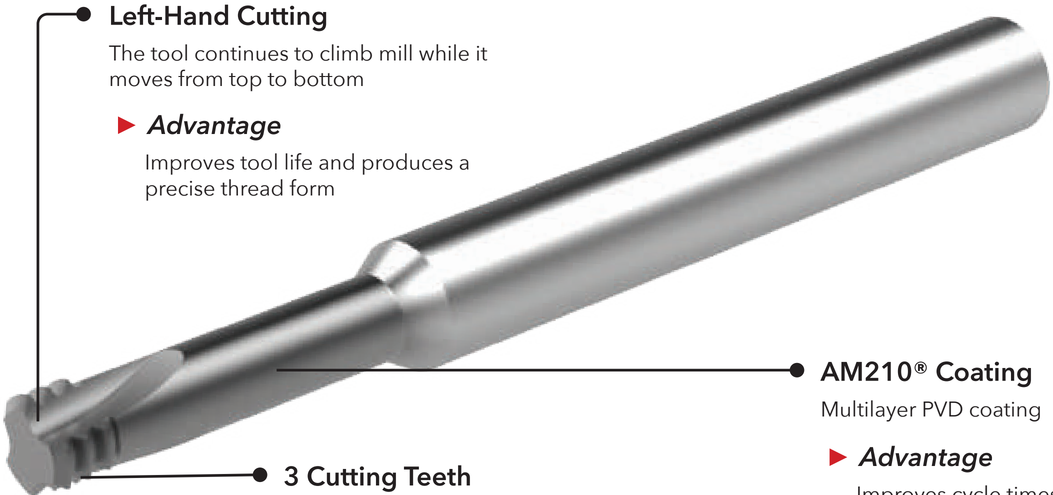


NOTE: See Section A92 of our product catalog for full AccuPort 432 product line information.

Solid Carbide Styles and Thread Forms

Straight BSW	**Helical** BSPP, NPS, NPSF, UN, ISO	**Taper Helical** BSPT, NPT, NPTF	**Helical (3-Tooth Style)** UN, ISO
 AccuThread® 856	 AccuThread® 856	 AccuThread® 856	 AccuThread® T3
 ThreadMills USA™ (coolant and non-coolant)	 ThreadMills USA™ (coolant and non-coolant)	 ThreadMills USA™ (coolant and non-coolant)	

AccuThread® T3



Left-Hand Cutting

The tool continues to climb mill while it moves from top to bottom

► ***Advantage***

Improves tool life and produces a precise thread form

AM210® Coating

Multilayer PVD coating

► ***Advantage***

Improves cycle times and tool life

3 Cutting Teeth

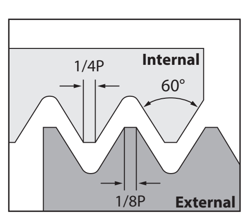
The tool cuts minimal threads at once and reduces side deflection

► ***Advantage***

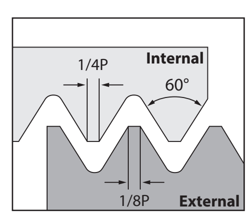
Cuts harder materials and produces deeper threads than a standard thread mill

Additional Information

- Available in UN and ISO thread forms
- Available in imperial and metric shanks
- Available in 2xD and 3xD lengths



UN Thread Form



ISO Thread Form

Thread Mill Pre-Drill Information

Formula	Metric	Imperial
Velocity	M/min = RPM • 0.003 • Cutter Diameter	SFM = RPM • 0.262 • Cutter Diameter
Speed	$\text{RPM} = \frac{(\text{M/min} \bullet 318.47)}{\text{Cutter Diameter}}$	$\text{RPM} = \frac{(\text{SFM} \bullet 3.82)}{\text{Cutter Diameter}}$
Linear Feed Rate (LFR)	mm/min = RPM • (mm/tooth • Number of Flutes)	IPM = RPM • (IPT • Number of Flutes)
Adjusted Feed Rate (AFR) ***See Note Below***	$\text{AFR} = \left(\frac{(\text{Major Diameter - Cutter Diameter})}{\text{Major Diameter}}\right) \bullet \text{LFR}$	

NOTE: The above formula on an internal thread program adjusts the linear feed rate to be applied to the outer diameter instead of the center of the cutting tool. If the feed rate is not adjusted, the excessive feed rate will cause the thread mill cutting edges to fail.

Example of an Internal Adjusted Feed Rate Calculation:

Free machining steel at 125 BHN with a M16x2 2B thread using ThreadMills USA™ solid carbide thread mill (TM16200) running at 221 M/min and 0.038 mm/tooth

STEP 1:	$\text{RPM} = \frac{(\text{M/min} \bullet 318.47)}{\text{Cutter Diameter}}$	$\text{RPM} = \frac{(221 \bullet 318.47)}{11.94}$	RPM = 5895
STEP 2:	LFR = RPM • (mm/tooth • Number of flutes)	LFR = 5895 • (0.038 • 4)	LFR = 896.04 mm/min
STEP 3:	$\text{AFR} = \left(\frac{(\text{Major Diameter - Cutter Diameter})}{\text{Major Diameter}}\right) \bullet \text{LFR}$	$\text{AFR} = \left(\frac{(16.00 - 11.94)}{16.00}\right) \bullet 896.04$	AFR = 227.37 mm/min

Unit Definitions

Velocity	**M/min** = Meters per Minute **SFM** = Surface Feet per Minute
Speed	**RPM** = Revolutions per Minute
Feed	**mm/rev** = millimeters per revolution **mm/tooth** = millimeters per tooth *also known as* millimeters per flute **IPR** = Inch per Revolution **IPT** = Inch per Tooth *also known as* Inch per Flute **mm/min** = millimeters per minute **IPM** = Inches per minute

XTP TAP TAS HPU APX 4TX REV OPN SSD ACP BTA WHL CRT ALV BRN THM

Thread Mill Calculations and Recommended Passes

Thread Mill Drill Calculation

Based on nominal tap drill diameter. Based on 0.003" or 0.075 mm probable mean oversize.

To calculate the percent of full thread for a given hole diameter:

IMPERIAL:

$$\text{\% of thread} = \text{\# of threads per inch} \bullet \frac{\text{Basic major diameter of thread} - \text{Drill hole size}}{0.0130}$$

METRIC:

$$\text{\% of thread} = \frac{76.96}{\text{Pitch (mm)}} \bullet [\text{Basic major diameter of thread} - \text{Drill hole size}]$$

Major Thread Diameter for # Drills

Drill #	Thread Diameter
# 2	0.086
# 3	0.099
# 4	0.112
# 5	0.125
# 6	0.132
# 8	0.164
# 10	0.190
# 12	0.216

Recommended Passes

NPT / NPTF / BSPT / API			
Pitch Size	Machinability: Easy	Average	Difficult
28	1	1	2
27	1	1	2
19	1	1	2
18	1	1	2
14	1	2	3
11.5	1	2	3
11	1	2	3
10	1	2	3
8	2	3	4

- 1 Pass
- 2 Passes
- 3 Passes
- 4 Passes

ISO			
Pitch Size	Machinability: Easy	Average	Difficult
0.40	1	1	2
0.45	1	1	2
0.50	1	1	2
0.70	1	1	2
0.75	1	1	2
0.80	1	1	2
1.00	1	1	2
1.25	1	2	3
1.50	1	2	3
1.75	1	2	3
2.00	1	2	3
2.50	2	3	4
3.00	2	3	4
3.50	2	3	4
4.00	2	3	4
4.50	2	3	4
5.00	2	3	4
6.00	2	3	4

UN / UNJ / BSPP / BSW / NPS / NPSF			
Pitch Size	Machinability: Easy	Average	Difficult
64	1	1	2
56	1	1	2
48	1	1	2
44	1	1	2
40	1	1	2
36	1	1	2
32	1	1	2
28	1	1	2
24	1	1	2
20	1	2	3
19	1	2	3
18	1	2	3
16	1	2	3
14	1	2	3
13	1	2	3
12	1	2	3
11	2	2	4
10	2	3	4
9	2	3	4
8	2	3	4
7	2	3	4
6	2	3	4

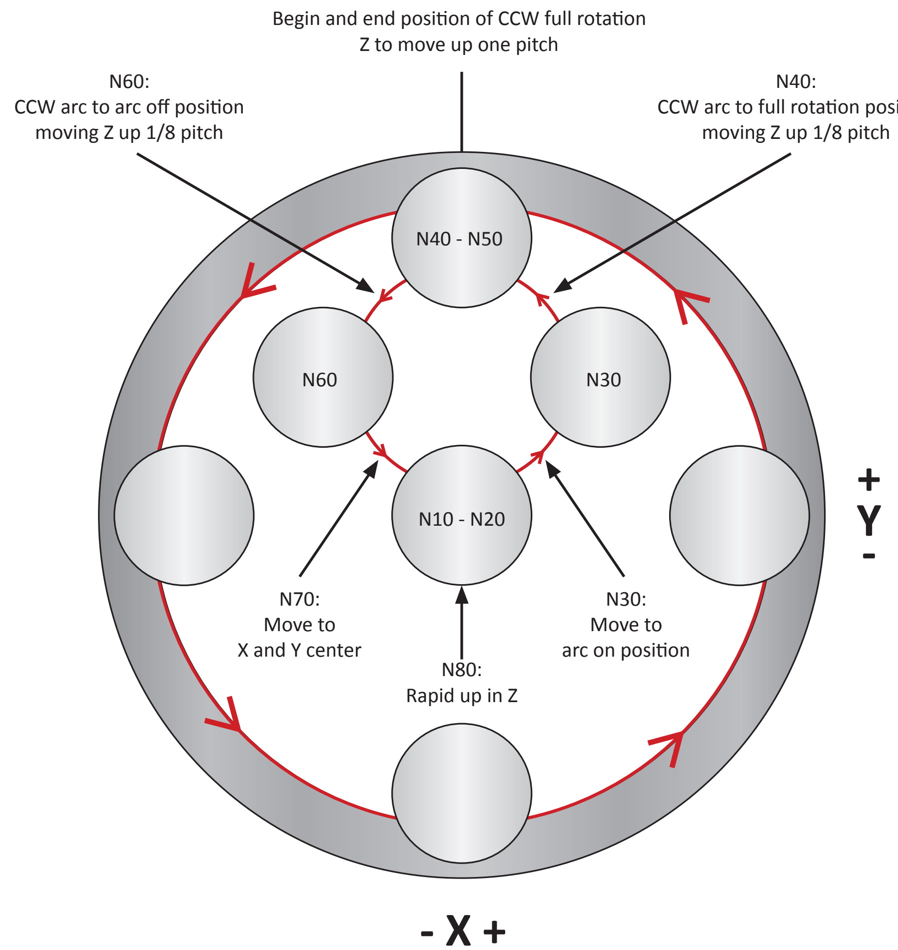
Thread Mill Programming Guide

What you need to know
- Thread milling can be easily accomplished with simple G code programming
- If your machine is capable of 3 axis (helical) interpolation, you can and **should** be thread milling
- Basic programming of a one pass thread mill can be achieved in 6 basic steps

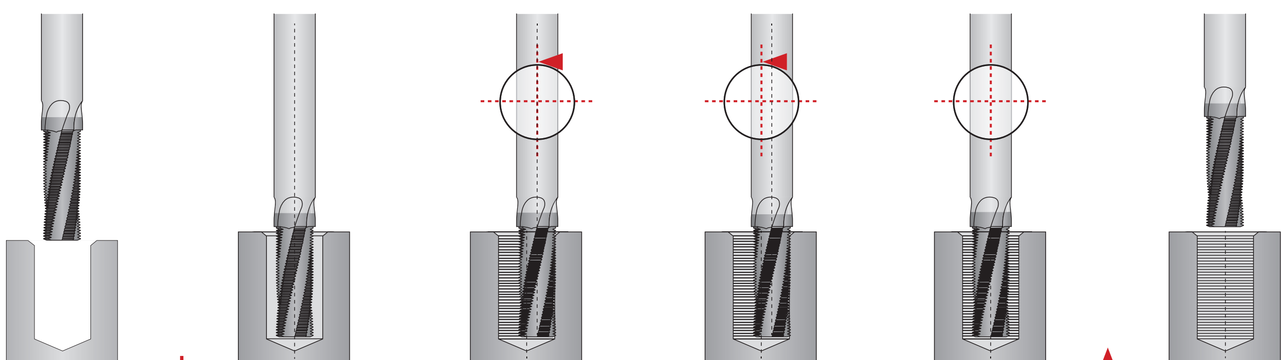
The following are examples of how to calculate and program a M16x2 right hand thread that will be 10mm deep produced in one pass

Major thread diameter	**16mm**	Major diameter of thread
Threads per inch		(only applies to imperial threads)
Length of thread	**10mm**	Desired length of cut
Velocity	**221 M/min**	Recommended velocity for material to be cut
Feed per flute	**0.038mm/tooth**	Recommended feed rate per cutting edge
Number of flutes	**4**	Number of flutes on tool to be used
Cutter diameter	**11.94mm**	Diameter of cutting tool
Using the information above, the values can be calculated:		
Pitch	**2.0mm**	Use 1/ threads per inch for imperial
Speed	**5895 RPM**	(318.47 • M/min) / cutter diameter or (SFM • 3.82) / cutter diameter
Linear feed	**896.04mm/min**	RPM • (Feed per flute • Number of flutes)
Feed rate for thread milling	**227.37mm/min**	((Major thread diameter - cutter diameter) / Major thread diameter) • Linear feed
Z-axis travel on arc on	**0.25mm**	(Pitch / 8)
Z-axis travel for full thread	**10.25mm**	(Pitch / 8) + Length of cut
Arc on/off	**1.015mm**	(Major thread diameter - cutter diameter) / 4
Full rotation value	**2.030mm**	(Major thread diameter - cutter diameter) / 2

Major thread diameter	16 mm	**Feed rate for thread milling**	227.37 mm/min	**Arc on/off value**	1.015 mm
Cutter diameter	11.94 mm	**Z axis depth for full thread**	10.25 mm	**Full rotation value**	2.030 mm
Length of thread	10.00 mm	**Z axis for arc on/off**	0.25 mm	**Pitch value**	2.00 mm



Step	Block	Code								Description
1	N10	S		5895	M03					Turn on spindle in the clockwise direction.
2	N20	G91	G01			Z -10.250			F 1136.25	Lower tool the depth required plus an additional 1/8 pitch for the arc on. Tool remains in the center of the hole.
3	N30	G41		X 1.015	Y 1.015		D1		F 681.75	Position the tool for the arc on motion and turn on the cutter compensation.
3	N40		G03	X -1.015	Y 1.015	Z 0.250	I -1.015	J0.000	F 227.37	Arc on to engage the tool to the major thread diameter while moving the tool up in the Z-axis 1/8 pitch. **NOTE**: X and Y is the end point. I and J is the center point of the arc.
4	N50		G03	X 0.000	Y 0.000	Z 2.000	I 0.000	J -2.030		Full pass. Interpolate the thread mill inside the major thread diameter while moving the tool up 1 pitch in the Z axis.
5	N60		G03	X -1.015	Y -1.015	Z 0.250	I 0.000	J -1.015	F 909.00	Arc off to disengage the tool from the major thread diameter while moving the tool up in the Z-axis 1/8 pitch.
5	N70	G40	G01	X 1.015	Y -1.015				F 1136.25	Return to center. Turn off cutter compensation and move the tool back to center.
6	N80		G00			Z 7.750				Return to top. Rapid the tool back to the top of the threaded hole.
6	N90	G90								Switch back to absolute positioning and rapid to a safe point in Z above part level (assumed to be 1 pitch above part level for demonstration purposes below).



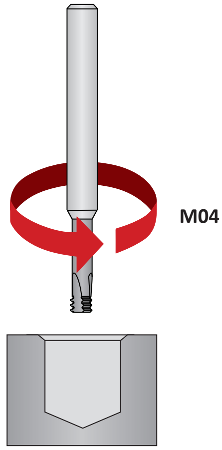
Step 1 N10	Step 2 N20	Step 3 N30 - N40	Step 4 N50	Step 5 N60 - N70	Step 6 N80 - N90
• Preparatory commands • Positioning above hole center and at hole level in Z • In absolute position mode	• Change to incremental • Feed to bottom of hole • Z axis depth for full thread	• Activate left cutter comp • Feed to arc on position • Arc to full rotation value while moving Z up 1/8 pitch • Z axis move for arc on	• One complete CCW rotation at full arc rotation value while moving Z up 1 pitch value	• CCW arc from full rotation value to the arc on/off value while moving Z up 1/8 pitch (Z axis move for arc off)	• Rapid up in Z

Technical Information

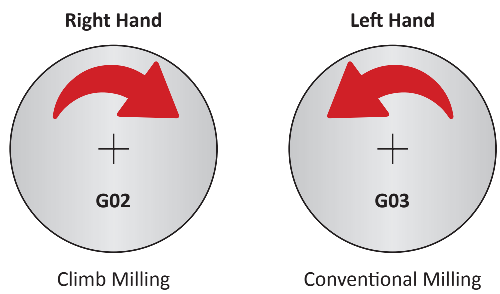
AccuThread® T3

Spindle Rotation

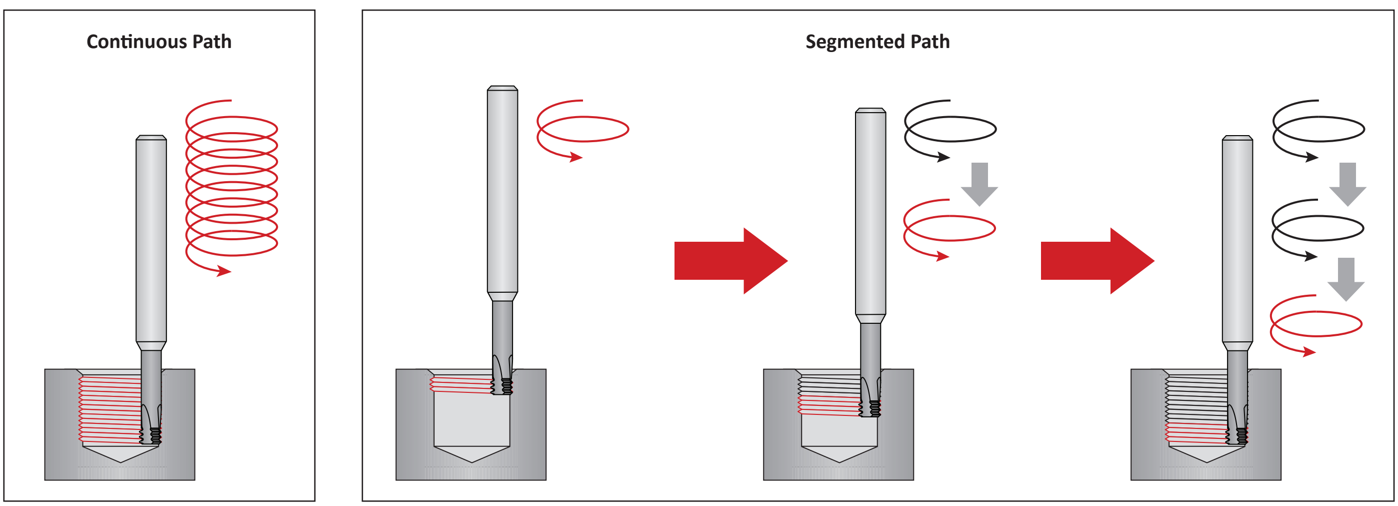
❗ **Tools are left-hand cutting.** The left-hand cut allows the tool to climb mill when creating a right hand thread with an AccuThread T3. Climb milling reduces deflection and heat generated during the cut.

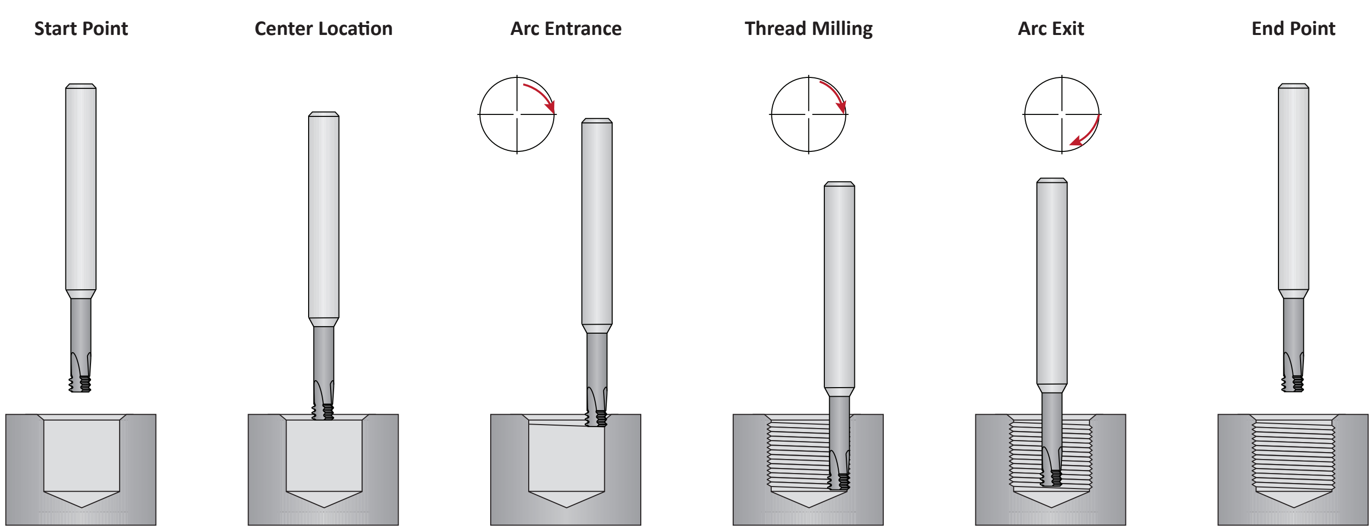


Direction of Helical Interpolation



Programming Z-Axis Cutting Path





XTP
TAP
TAS
HPU
APX
4TX
REV
OPN
SSD
ACP
BTA
WHL
CRT
ALV
BRN
THM

Thread Mill Troubleshooting Guide

Causes		Problem: Thread mill is showing accelerated or excessive wear	Cutting edges are chipping	Thread mill is breaking in the first hole of part	Thread mill is creating excessive chatter	Out of round thread is produced	Bell-mouthed thread form (small at bottom, big at top)	Part rejection because of rough flank finish	Steps in thread profile	Gauge difference from part to part	Machine not making correct paths to create thread profile	Control not accepting the program
Catalog	Incorrect tool selection			1	1							
Catalog	Incorrect speed and feed selection	2, 3	2, 3		2, 3			2, 3				
Speed and Feed	RPM too high	5										
Speed and Feed	RPM too low				4		4	4				
Speed and Feed	Machine tool specifications restrict RPMs			5, 19								
Speed and Feed	Feed rate too high		7	7			7	7	7			
Speed and Feed	Feed rate too low	6										
Speed and Feed	Incorrect adjusted feed rate adjustment ratio			12								
Speed and Feed	Machine tool specification restricts feed rate					7, 19						
Speed and Feed	Ramp-in is programmed as an axial move			20					20			
Tool	Thread mill moved or slipped in its holding device	13	13	13	13			13	13			
Tool	Tool is sticking out of the holder too far	15	15	15	15			15	15	15		
Tool	Runout between thread mill and holder				10			10				
Tool	Incorrect coating creating built up edge	8, 17								8, 17		
Tool	Helix angle too low				9			9				
Tool	Excessive thread mill wear								11	11		
Tool	Excessive tool pressure	7, 11, 14					7, 11, 14					
Machine	Workpiece moving in its fixturing	16	16	16	16			16		16		
Machine	Insufficient coolant pressure or flow	17	17									
Machine	Lack of machine rigidity	16	16		16		16	16				
Programming	Incorrect number of passes			22			22					
Programming	Incorrect program variables			18, 26							18, 26	
Programming	Did not account for X/Y radial moves for tapered threads										24, 26	
Programming	Incorrect cutter compensation variables			23, 26								23, 26
Programming	Helical interpolation option not on machine or turned off										21, 26	21, 26
Programming	Machine tool control is not formatted to standard EIA/ASCII/ISO Code											25, 26

Troubleshooting Solutions

1. Refer to catalog to ensure proper tool selection.
2. Verify the correct speed was selected from the catalog speed and feed chart.
3. Verify the correct feed rate was selected from the catalog speed and feed chart.
4. Increase the spindle speed (RPM).
5. Decrease the spindle speed (RPM).
6. Increase feed per tooth.
7. Decrease feed per tooth.
8. Investigate other coatings.
9. Increase the tool helix.
10. Gauge runout between thread mill and tool holder.
11. Perform tool change at quicker intervals.
12. Adjust the feed rate ratio properly to the correct actual penetration rate for internal threads. Refer to speed and feed pages for formula.
13. Use hydraulic clamping chuck.
14. Check the tool for excessive wear. Beginning threads will wear the fastest.
15. Make the amount of overhang in the holding device as short as possible.
16. Verify the workpiece is properly clamped. Retighten or increase stability if necessary.
17. Increase the coolant flow and volume.
18. Check the milling program variables, especially the positive or negative value associated with I and J values.
19. Make sure the machine has the appropriate axis and path speed capabilities.
20. Make sure the thread mill is arcing in the major diameter instead of making a radial move.
21. Make sure the machine tool has a helical interpolation option that is on.
22. Increase the number of thread mill passes.
23. Make sure the cutter compensation variables are input into the G41 program line.
24. Adjust the program for pipe tap threads to taper out on diameter in X/Y directions to create proper form.
25. Request information from the machine tool builder regarding its programming formats.
26. Scan and email a copy of your program to the Application Engineering department at **appeng@alliedmachine.com**.

Guaranteed Test / Demo Application Form

Distributor PO #	

The following must be filled out completely before your test will be considered

IMPORTANT: For processing, send Purchase Order to your Allied Field Sales Engineer (FSE). Please clearly mark the paperwork as "Test Order."

Distributor Information

Company Name: ______________________

Contact: ______________________

Account Number: ______________________

Phone: ______________________

Email: ______________________

End User Information

Company Name: ______________________

Contact: ______________________

Industry: ______________________

Phone: ______________________

Email: ______________________

Current Process

List all tooling, coatings, substrates, speeds and feeds, tool life, and any problems you are experiencing

____

____

Test Objective

List what would make this a successful test (i.e. penetration rate, finish, tool life, hole size, etc.)

____

____

Application Information

Hole Diameter: ________ in/mm

Tolerance: ________

Material: ________ (4150 / A36 / Cast Iron / etc.)

Pre-existing Diameter: ________ in/mm

Depth of Cut: ________ in/mm

Hardness: ________ (BHN / Rc)

Required Finish: ________ RMS

State: ________ (Casting / Hot rolled / Forging)

Machine Information

Machine Type: ________ (Lathe / Screw machine / Machine center / etc.)

Builder: ________ (Haas, Mori Seiki, etc.)

Model #: ________

Shank Required: ________ (CAT50 / Morse taper, etc.)

Power: ________ HP/KW

Thrust: ________ lbs/N

Rigidity:
- ☐ Excellent
- ☐ Good
- ☐ Poor

Orientation:
- ☐ Vertical
- ☐ Horizontal

Tool Rotating:
- ☐ Yes
- ☐ No

Coolant Information

Coolant Delivery: ________ (Through tool / Flood)

Coolant Pressure: ________ PSI / bar

Coolant Type: ________ (Air mist, oil, synthetic, water soluble, etc.)

Coolant Volume: ________ GPM / LPM

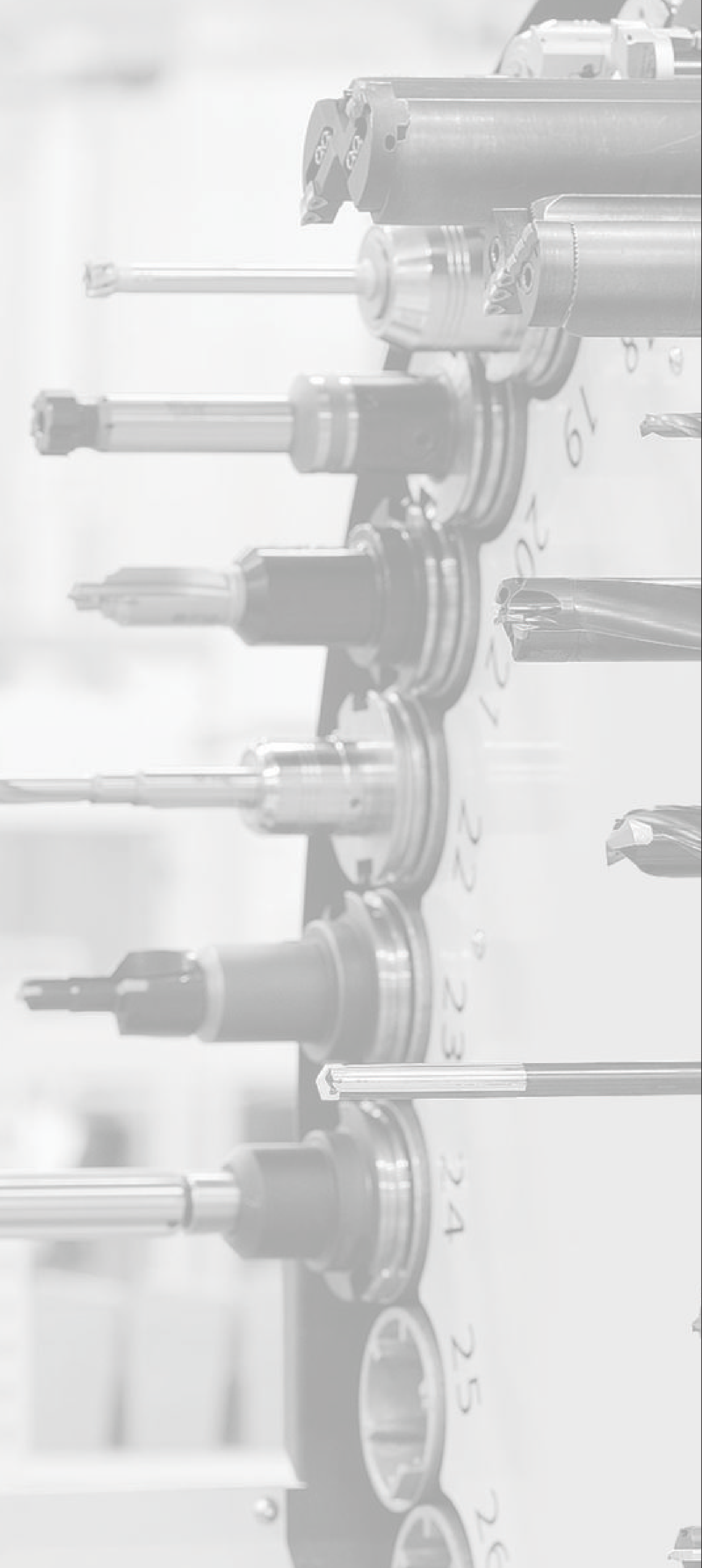
Requested Tooling

QTY	Item Number	QTY	Item Number

ALLIED MACHINE & ENGINEERING

Allied Machine & Engineering
120 Deeds Drive
Dover, OH 44622

Telephone: (330) 343-4283
Toll Free USA & Canada: (800) 321-5537
Fax: (330) 602-3400
Email: info@alliedmachine.com



ALLIED MACHINE
& ENGINEERING

Warranty Information

Allied Machine & Engineering ("Allied Machine") warrants to original equipment manufacturers, distributors, industrial and commercial users of its products for one year from the original date of sale that each new product manufactured or supplied by Allied Machine shall be free from defects in material and workmanship.

Allied Machine's sole and exclusive obligation under this warranty is limited to, at its option, without additional charge, replacing or repairing this product or issuing a credit. For this warranty to be applied, the product must be returned freight prepaid to the plant designated by an Allied Machine representative and which, upon inspection, is determined by Allied Machine to be defective in material and workmanship.

Complete information as to operating conditions, machine, setup, and the application of cutting fluid should accompany any product returned for inspection. This warranty shall not apply to any Allied Machine products which have been subjected to misuse, abuse, improper operating conditions, improper machine setup or improper application of cutting fluid or which have been repaired or altered if such repair or alteration, in the judgement of Allied Machine, would adversely affect the performance of the product.

THIS WARRANTY IS IN LIEU OF ALL OTHER WARRANTIES, EXPRESS OR IMPLIED, INCLUDING ANY IMPLIED WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. Allied Machine shall have no liability or responsibility for any claim, whether in contract, tort or otherwise, for any loss or damage arising out of, connected with, or resulting from the manufacture, sale, delivery or use of any product sold hereunder, in excess of the cost of replacement or repair as provided herein.

Allied Machine shall not be liable in contract or in tort (including, without limitation, negligence, strict liability or otherwise) for economic losses of any kind or for any special, incidental, indirect, consequential, punitive or exemplary damages arising in any way out of the performance of, or failure to perform this agreement.

ALL PRICES, DELIVERIES, DESIGNS, AND MATERIALS ARE SUBJECT TO CHANGE WITHOUT NOTICE.

Certified
DQS
Management System

Allied Machine & Engineering is registered to ISO 9001:2015 by DQS

United States

Allied Machine & Engineering
120 Deeds Drive
Dover OH 44622
United States

Phone:
+1.330.343.4283

Fax:
+1.330.602.3400

Toll Free USA and Canada:
800.321.5537

Toll Free USA and Canada:
800.223.5140

Allied Machine & Engineering
485 W Third Street
Dover OH 44622
United States

Phone:
+1.330.343.4283

Fax:
+1.330.364.7666
(Engineering Dept.)

Toll Free USA and Canada:
800.321.5537

Europe

Allied Machine & Engineering Co. (Europe) Ltd.
93 Vantage Point
Pensnett Estate
Kingswinford
West Midlands
DY6 7FR England

Phone:
+44 (0) 1384.400900

Wohlhaupter GmbH
Maybachstrasse 4
Postfach 1264
72636 Frickenhausen
Germany

Phone:
+49 (0) 7022.408.0

Fax:
+49 (0) 7022.408.212

Asia

Wohlhaupter India Pvt. Ltd.
B-23, 3rd Floor
B Block Community Centre
Janakpuri, New Delhi - 110058
India

Phone:
+91 (0) 11.41827044

Your local Allied Machine representative:

www.alliedmachine.com

Allied Machine & Engineering is registered to **ISO 9001:2015** by DQS

ALLIED MACHINE & ENGINEERING

Available Online Only
Publish Date: 2023