HOLEMAKING

DRILLING:

- Drills are used to make holes in a variety of operations from maintenance & repair to high volume precision holemaking.
- Many factors influence the selection process when choosing a drill, including drill style, performance level, tool and workpiece material and drill point angle.
- Coatings or surface treatments provide improved tool performance and increase in tool life.

DRILL STYLES

- Straight Shank: shank diameter matches the cutting diameter. These are the most common drills and are effective on both ferrous and non-ferrous materials. Jobbers length and screw machine length are widely used.

SPECIALTY STRAIGHT SHANK DRILLS

- **Parabolic**: used for deep hole drilling and reduce the need for pecking.

- **Slow Spiral:** used for long chipping materials like aluminum and copper.

- **Fast Spiral:** used on softer materials where chip evacuation is important.

- **Half Round:** used in deep hole drilling on soft materials like brass & rubber.

- **Straight Flute:** used on hard materials.

- **Core:** used to enlarge existing holes and are also available in taper shank.

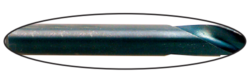
OTHER HOLE MAKING TOOLS

- **Annular Cutters:** also called a core drill, core cutter, broach cutter, trepanning drill hole saw, or cup-type cutter is a hollow cutting tool, with multiple cutting edges used to make/drill holes in ferrous and non-ferrous metals. It cuts an annular groove at the periphery of the hole and leaves a solid core or slug at the hole center. It can also be used as a substitute for slower, more expensive and less efficient twist drills and hole saws. The function of annular cutters is similar to a hole saw, but differs in geometry and material. Annular cutters are faster, easier and are more accurate than the conventional twist drills or drill bits.

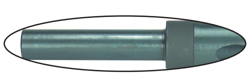
- **Hole Saws:** also known as a hole cutter, is a saw blade of annular shape, whose annular kerf creates a hole in the workpiece without having to cut up the core material. It is used in a drill. Hole saws typically have a pilot drill bit at their center to keep the saw teeth from walking.

SHANK OPTIONS

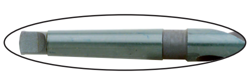
Non-cutting end that is held by a machine tool accessory to drive or hold the drill bit.



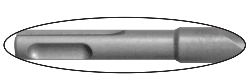
- **Straight Shank:** most usual style for modern drill bits. Shank size is always made the same diameter as the drill bit.



- **Reduced Straight Shank:** drill shank diameter is smaller than the drill bit size (cutting diameter). They can drill holes that are larger than the capacity of the drill chuck. Larger cutting diameters than straight shank's drill diameters, so that drill chucks can be used to drill large holes.



- **Taper (Morse Taper):** allows the bit to be mounted directly into the spindle of a drill, lathe tailstock, or (with the use of adapters) into the spindle of milling machines. Has very accurate centering.



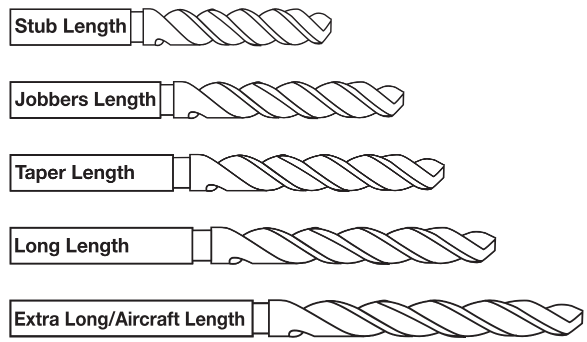
- **SDS:** has the advantage of a simple spring-loaded chuck, so that bits can be chucked with a simple and quick hand action. Further, the shank and chuck are uniquely suited to hammer drilling in stone and concrete.

Phone: **1.800.221.0270** | Fax: **1.800.722.0703** | Web: **www.travers.com** | Email: **sales@travers.com**

DRILL LENGTHS

In general, the shorter the drill, the better the performance. The shortest length provides the best rigidity thus creating truer holes.

- **Stub Length:** short length drill bits or screw machine drill bits are shorter in flute and overall length than jobbers length drills. Reduces deflection and breakage and gives maximum rigidity and accuracy. For hand and machine drilling.
- **Jobbers Length:** jobbers length drills are the most common type of drill. The length of the flutes is either four or five times the diameter of the drill. For general purpose or heavy duty drilling.
- **Taper Length:** longer flute and overall length than jobbers drill. For drilling deep holes.
- **Long Length:** for drilling deep and hard to reach spots.
- **Extra Long/Aircraft Length:** for drilling extra deep and hard to reach spots.



DRILL POINT ANGLES

Selection of drill point angles affect tool performance. Flat points, (larger point angles) offer better cutting action & more aggressive angles create torque. Sharper points (smaller point angles) are better for soft, non-ferrous materials.

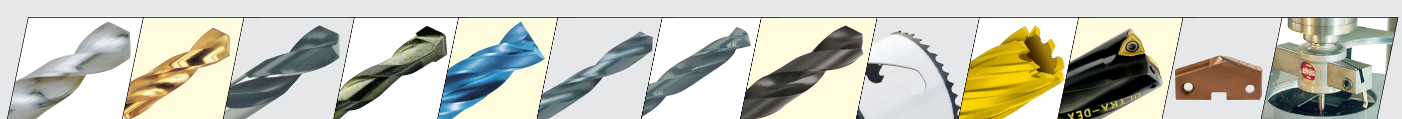
- 118°: General purpose
- 130°: For mild steel
- 135°: For harder materials
- 125°: For soft non-ferrous materials
- 134°: For mild steel
- 140°: For hard materials and high performance
- 150°: For hard materials and high performance

PERFORMANCE LEVELS

Combination of the drill geometry, tool material and coating helps drive the level of performance.

- **Economy:** used for short run jobs and provides the lowest price point option.
- **General Purpose:** generally preferred for the machining of non-ferrous and non-metallic materials and alloys.
- **Deep Hole:** for general purpose and heavy duty drilling operations.
- **High Performance:** meets the tough demands of production level manufacturing. Longer life and recommended for increased speeds and feeds when drilling aluminum, cast iron and other easy to machine materials.
- **Heavy Duty:** for drilling a wide variety of ferrous materials including steel, cast steel, cast iron, stainless steels and other alloyed or non-alloyed ferrous metals.

DRILL SELECTION GUIDE:



Size Range	.0156" To .2598"	1/64" To 3/4"	33/64" To 2"	1/8" To 1"	1mm To 3/4"	.0135" To 1-3/4"	.0440" To 1/2"	1/8" To 4"	9/16" To 6"	1/4" To 4"	.484" To 2-3/8"	7/16" To 6"	20mm To 300mm
Drill Style	Micro Drills	Jobbers & Screw Machine	Silver & Deming	Masonry	Flat Bottom	Taper Length	Aircraft	Taper Shank	Hole Saws	Annular Cutters	Indexable Drilling	Spade Drills	Holemaking Systems
Tool Material	Cobalt Solid Carbide	H.S.S. Cobalt Solid Carbide	H.S.S. Cobalt Carbide Tipped	Carbide Tipped	Solid Carbide	H.S.S. Cobalt	H.S.S. Cobalt	H.S.S. Cobalt Carbide Tipped	H.S.S. Bi-Metal Carbide Tipped	H.S.S. Cobalt Carbide Tipped	Carbide Inserts Steel Bodies	H.S.S. Cobalt	H.S.S.
TRAVERS Portfolio	•	•	•	•	•	•	•	•	•	•	•	•	•

DRILL FINISH/COATINGS

Coating or surface treatments have a direct effect on tool performance and tool life. They also build a barrier between the drill and workpiece.

- **AG Coated:** TiAlN Multilayer Coating – Outstanding thermal and chemical resistance properties increases the stability of the cutting edges & improved chip flow.
- **AlTiN:** Aluminum Titanium Nitride Coating – Ideal for high temperature cutting operations in titanium and nickel alloys, stainless steel, steels and cast irons. It creates a hard aluminum oxide layer as temperature increases, insulating the tool and transferring heat into the cutting chips.
- **Black & Bronze Oxide Coated:** Reduces friction and allows for higher speeds.
- **Bright Finish:** Surface is untreated. Use for general purpose drilling of most metals, wood and plastic.
- **Bronze Oxide Coated:** Reduces friction & allows for higher speeds. Known appearance color for cobalt.
- **SG Coated:** Silicone Multilayer Coating – Use for high performance drilling of carbon steels, stainless steels, aluminum and various high-temp alloys.
- **Steam/Black Oxide:** An inexpensive black coating. A black oxide coating provides heat resistance and lubricity, as well as corrosion resistance. These result in a longer bit life than possible for the typical uncoated high speed steel bits. For drilling of ferrous metals such as steel, stainless steel and cast iron.
- **TiAlN Coated:** Super Life Titanium Aluminum Nitride – Dark Violet/ Black - considered superior to TiCN and can extend tool life five or more times. More stable at higher temperatures than TiN or TiCN. Increases machining speeds by 75% to 100%.
- **TiN Coated:** Long Life Titanium Nitride - Bright Gold - is a very hard ceramic material, and when used to coat a high-speed steel bit can extend the cutting life by three or more times. Increases machining speeds by 25% to 30%.
- **TiCN Coated:** Extra-Life Titanium Carbonitride – Violet/ Blue-Gray - superior to TiN and more wear resistant; increases machining speeds by 35% to 50%.

TOOL MATERIAL

Choice of tool material is driven by the workpiece material, number of holes desired along with the life of the drill.

- **High Speed Steel:** economical choice for general applications in most materials including steel, cast iron and forgings. Provides good tool life and efficiency.
- **Cobalt:** M35, M42, better than high speed steel, they hold their hardness at much higher temperatures and are well suited for harder to machine materials and stainless steels.
- **Solid Carbide:** resists wear better than both high speed steel and cobalt steel. Has a better degree of accuracy for drilling very hard materials such as stainless steel, titanium alloys and nickel alloys.
- **Carbide Tipped:** same benefits as solid carbide for a fraction of the cost.

THE BENEFITS OF COOLANT FED DRILLS

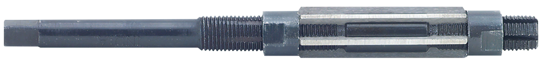
Using coolant through tooling for holemaking reduces the tool's core temperature, while it simultaneously improves lubricity and chip evacuation. This not only effectively improves tool performance and extends tool life, but it also reduces the cost per hole. Workpiece results are improved with truer holes and tighter tolerances as well.

Reduced Cost:
The coolant through tool allows for high penetration rates, and an increase in feeds and speeds while reducing or eliminating the pecking cycle. This results in decreased cycle time and lower cost per hole.

Extended Tool Life:
The cost per hole is further decreased by using a tool that won't need replacing as often. With reduced temperatures for both the drill and the work-piece, as well as increased lubricity, the life of the tool is extended because coolant is delivered directly to the cutting edge—therefore cooling the area and aiding in chip evacuation.

Better Quality Product:
Coolant drilling creates rounder and straighter holes and smoother finishes as the coolant washes chips away from the drill point. These chips would otherwise cause binding or marking, in addition to damaging the workpiece, so this process reduces the need for finishing operations such as reaming, honing or boring.

TYPES OF REAMERS:



ADJUSTABLE BLADES INCREASE REAMING CAPACITY

Adjustable Blade Reamers: For hand resizing of drilled or bored holes to precise diameters.

Ball Joint Taper Reamers: For deburring or enlarging tapered holes for tie-rod ends or ball joint studs. When using to fabricate new parts, drill holes as close to size as possible before reaming them. Handles can be removed.

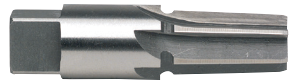
Bridge Reamers: Designed to enlarge and align holes for rivets. Primarily used in steel construction using I-beams. Shanks come in different styles for square-drive impact wrench sockets, drill presses and portable electric drills.

Chucking Reamers: Created for usage in screw machines, lathes and drill presses. Most frequently used style for reaming holes to a tight tolerance. Comes in straight flute, LHS/LHC, RHS/RHC and expansion styles in jobber and standard lengths.

Construction Taper Reamers: Use to align holes prior to bolt or rivet installation. This design provides the finest hole finish.

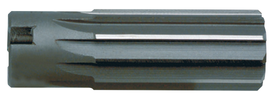
Expansion Reamers: Can be reground many times due to the nature of it being built with an adjustable screw to alter the size of the cutting head.

Hand Reamers: Contains a square at the end of the shank which allows the user to rotate the reamer with a standard tap wrench or an adjustable wrench, similar to a hand tap. Constructed with a tapered cutting head to assist the user in pushing the tool into the workpiece.



Pipe Reamers: Commonly used to prepare a hole for tapping with a taper pipe tap. Constructed with a square end on the shank to assist the user in turning the reamer with a tap wrench or an adjustable wrench.

Repairman's Reamers: Built for enlarging holes in sheet metal and for the removal of burrs in conduit and pipe. Handles can be removed.



Shell Reamers: Similar to the cutting portion of a chucking reamer, and supplied without a shank. Taper shank
arbor or straight shank is used in conjunction with the shell reamer, used as a complete cutting tool.

Taper Car Reamers: Used to align existing holes prior to bolt or rivet operations.

REAMING RECOMMENDATIONS:

Workpiece hardness and machinability must be considered when setting machine speed. The feed rate plays an important part in the life expectancy of a tool and the hole finsih you wish to attain. Improper feed rate can cause excessive tool wear as well as an inadequate hole finish.

To elminate chatter, slow cutting speed and increase feed appropriately. Stock removal on roughing operations should not exceed 2% to 4% of tool diameter in most cases.

In all reaming operations, use constant-flow coolants. Soluble oil is effective for most metals; however, sulphur-based oils are recommended for stainless and certain alloy steels.

REAMED HOLE

Recommended Stock Removal

Reamer Diameter	Removal (Inches)
Up to 1/16 Incl.	.003 to .005
Over 1/16 to 1/8 Incl.	.004 to .008
Over 1/8 to 1/4 Incl.	.006 to .012
Over 1/4 to 3/8 Incl.	.008 to .014
Over 3/8 to 1/2 Incl.	.010 to .015
Over 1/2 to 3/4 Incl.	.012 to .018

Recommended Lubricants

Material	Lubricant
Steel harder than Rockwell C50	Light Oil
Steel softer than Rockwell C50	Light Oil For Good Finishes or Soluble Oil and Water
Cast Iron & Malleable Iron	Soluble Oil and Water
Non-Ferrous Materials	Soluble Oil and Water

All Dimensions in Inches

Recommended Feeds

Material	Feed in Inches Per Revolution
Steels	
Rockwell C50 or Harder	.002 to .004
Rockwell C30 to 50	.004 to .008
Cast Iron & Malleable Iron	.005 to .012
Non-Ferrous Materials	.005 to .012

Recommended Speeds

Material	Speed in Surface Feet Per Minute
Steel (All Types)	
Rockwell C60 or Harder	8 - 12
Rockwell C50 to 60	15 - 30
Rockwell C40 to 50	20 - 40
Rockwell C30 to 40	35 - 65
Under Rockwell C30	60 - 90
Cast Iron and Malleable Iron	50 - 85
Non-Ferrous	
Aluminum, Brass, Bronze, Copper, Fiber, Plastic, Hard Rubber, etc.	90 - 175

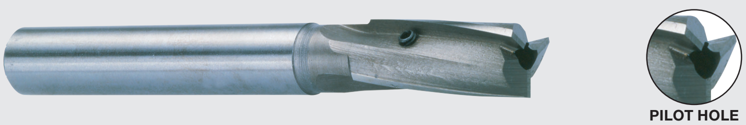
7 STEPS TO OPTIMAL REAMING RESULTS

For the best reaming results, prepare your holes with just the right amount of stock left in them. Too little, and the reamer will rub, leaving you with more wear and less diameter than expected. Too much and you may have to repeat the process. A precise balance of stock in your holes will give your reamer the best surfaces for optimal performance.

1. Start by checking to see if cutting needs to be done on the bevel lead or both the bevel and taper leads. A hand reamer offers a longer taper lead than the 45° bevel on a machine reamer, and will cut on both while offering different alignment solutions.
2. After selecting the optimal reamer option for your materials, ensure that your pre-drilled holes are the correct diameter. Now you can set the correct speeds and feed for your application.
3. Keep your workpieces rigid, and ensure there is zero play in your spindle machine set-ups.
4. Using higher quality chucks to hold your straight shank reamer can help prevent slippage. Slippage during automatic feeder operation can cause damage and breakage to your reamers.
5. Always check for fit tolerances to prevent misalignments between your reamer shank options and the sleeves, pockets or machine spindles you will be driving into.
6. Reaming doesn't require heavy cutting fluids for optimal performance, so utilizing soluble oils with 40-to-1 dilution should suffice for most applications. If dry machining gray iron, air blasting can improve your surface preparation results.
7. Check concentricity between your high-precision centers regularly, and regrind your bevel leads when necessary. Keeping your reamers sharp can maintain accuracy and hole quality while improving tool life.

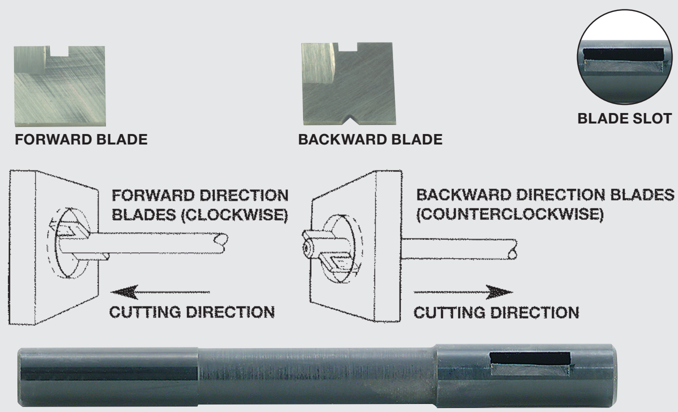
COUNTERBORE TYPES

Counterbores hide the head of a screw or bolt by enlarging the top portion of an existing hole. If the depth is shallow, it is known as spot-facing.



Interchangeable Pilot: This style offers greater flexibility to the end user when selecting a pilot size. By switching the pilot, a single tool can be utilized in many different hole diameters. Pilot simply slides into the built-in pilot hole in the counterbore. Pilots are not included with interchangeable type counterbores.

Solid Cap Screw: One piece construction includes the pilot thereby providing a very accurate hole.



Inserted Blade: Unique design allows for forward and backward cutting. Insertable and replaceable forward and backward blades can be resharpened.

COUNTERBORING TROUBLESHOOTING

PROBLEM	SOLUTION
Excessive cutting edge wear or rough finish in ductile or free machining materials:	Increasing feed or reducing speeds can extend the life of your cutting tools.
Excessive cutting edge wear due to rough cutting edge:	Lightly hone your cutting edges with a fine grit diamond tool for smoother cutting.
Excessive cutting edge wear from insufficient coolant:	Double-check your coolant types and increase flow accordingly.
Excessive chipping from insufficient chip removal:	Reduce flute space by using larger diameter tools with fewer flutes.
Excessive chipping from work-hardened metals:	An increase in coolant flow can improve recutting for tough work-hardened materials.
Excessive chipping from worn tool holders:	Increasing the rigidity settings in your set-up can reduce any excessive vibration.
Short tool life due to excessive cratering:	Increase speeds or decrease feeds to extend your tool life.
Short Tool Life due to abrasive materials:	Extend tool life with a combination of decreased speeds, increased feeds and increased coolant flow.
Short Tool Life due to hard materials:	Reduce your speeds for increased rigidity and support.
Short Tool Life due to insufficient chip room:	Use larger diameter-sized tools with fewer flutes and less flute space.
Short Tool Life due to delayed resharpening:	Resharpen counterbores to original geometries for extended tooling life.
Glazed finish due to light feed:	Increase your feed for a more well-rounded finish.
Glazed or rough finish due to dull cutting edge:	Resharpen counterbores to original geometries for a more well-rounded finish.
Glazed finish due to insufficient clearance:	Resharpen your counterbores for additional clearance.
Chattering due to insufficient horsepower:	A smaller diameter tool with fewer flutes can reduce chattering when used with maintained speeds.
Chattering due to vibration:	Resharpen your counterbores for additional clearance to decrease vibration.