UNITurn & UNIMill
The cool tool®

PRODUCT INSTRUCTIONS

TABLE OF CONTENTS

PART NO.	PRODUCT NAME
1041	3-Jaw Chuck
1044	4-Jaw Chuck
1074	Steady Rest
1075	4-Jaw Self Centering Chuck
1090	Follower Rest
1160	WW Collets and Sets
1185	Vertical Milling Table
1201	Adjustable Tailstock Tool Holders
1220	Tailstock Spindle Extender
1270	Compound Slide
1291	Riser Blocks
2085	WW Collet Adapter
2090	Clockmaker´s Arbors
2110	W.R. Smith T-Rest
2200	Radius Cutting Attachment
2250	Quick-Change Toolpost and Tool holders
3001	Power Feed
3002	Cut-Off Tool and Holder
3004	Knurling Tool and Holder
3016	Rear Mounting Block
3038	Wood Tool Rest
3050	Vertical Milling Column
3052	Flycutter (and #3065, Slitting Saw Holder)
3054	Boring Head
3055	Morse #1 Blank
3060	Milling Collets and End Mill Holders
3100	Thread Cutting Attachment
3200	Indexing Attachment
3420	2" and 2½" Resettable Handwheels
3551	Mill Vise
3700	4" Precision Rotary Table
3701	Right Angle Attachment
4335	10,000 rpm Pulley Set
4360	Chip Guard
6100	Horizontal Milling Conversion
-----	Grinding Your Own Lathe Tools

The cool tool®

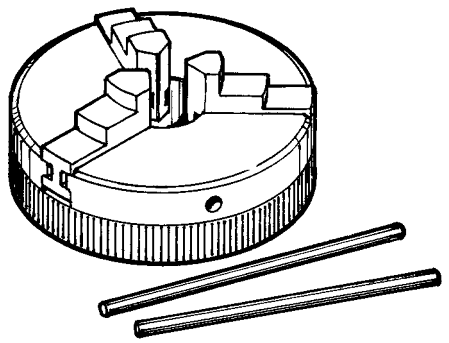
www.thecooltool.com

Modellbauwerkzeuge & Präzisionsmaschinen G.m.b.H.

Modelmaking & Precision Tools Ltd. Vienna / Austria

Fabriksgasse 15, A-2340 Mödling   info@thecooltool.com

phone: +43-2236-892 666   fax: +43-2236-892666-18



UNITurn & UNIMill
The cool tool

3-JAW CHUCKS

P/N 1040 (3.125") and P/N 1041 (2.5")

Three-Jaw Chucks are designed so that all three jaws move together and automatically center round or hexagonal parts or stock to within a few thousandths of an inch. These chucks provide the quickest and easiest way of holding work in the lathe.

The UNITurn Chuck is designed so that it can be used to clamp externally on bar stock or internally on tube stock. The P/N 1041 Chuck is designed to grip from 3/32" (2 mm) to 1-3/16" (30 mm) diameter stock with the jaws in the normal position. The P/N 1040 Chuck handles stock up to 1-1/2" (38 mm) in diameter. For larger diameter work, the jaws must be reversed (See Figure 2). The reversible jaws can grip to 2-1/4" (56.0 mm) for the P/N 1041 Chuck and up to 2.75" (70 mm) for the P/N 1040 Chuck. The chucks have a .687" (17 mm) diameter through hole with a 3/4"-16 thread.

Due to the nature of the design of a 3-jaw chuck, it cannot be expected to run perfectly true. Even 3-jaw chucks costing five times more than the one made for this lathe will have a 0.002" to 0.003" runout. If perfect accuracy is desired in a particular operation, the use of a 4-jaw chuck or a collet is recommended. Both are available for your UNITurn Lathe.

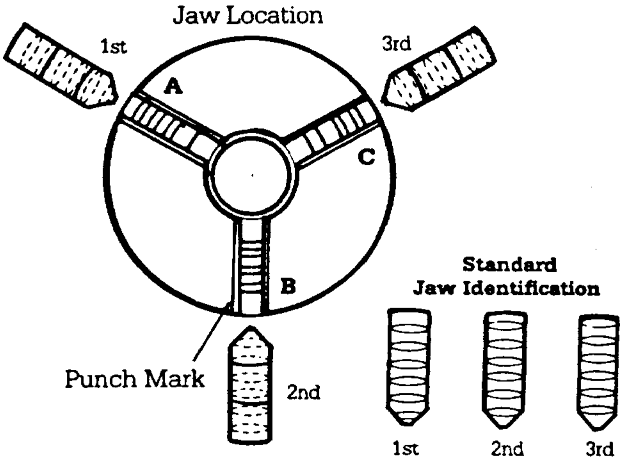


FIGURE 1—Three-Jaw Chuck, standard jaw locations.

NOTE: DO NOT TURN THE LATHE SPINDLE ON UNTIL THE CHUCK IS TIGHTENED. The acceleration of the spindle can cause the scroll to open the chuck jaws if not tightened!

To prevent permanent damage, finished, turned or drawn stock should only be held with this chuck. For rough castings, etc., use the 4-jaw chuck.

DO NOT OVERTIGHTEN THE CHUCK. Use only moderate pressure with the Tommy Bars supplied.

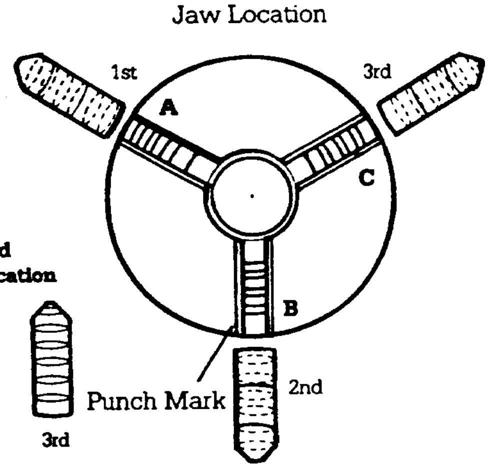


FIGURE 2— Reversing the Chuck Jaws.

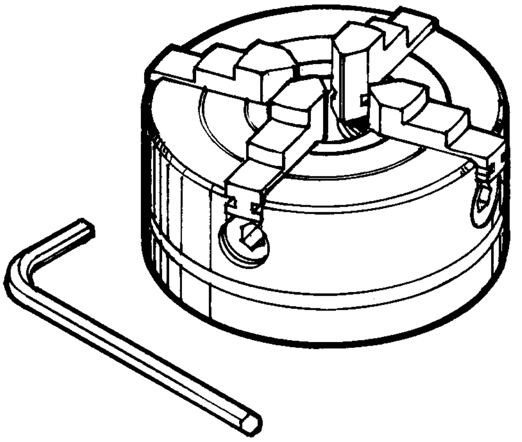
NOTE: Always start with position "A".

To reverse the chuck jaws, rotate the knurled scroll until the jaws can be removed. They can be easily identified by the location of the teeth to the end of the jaw (See Figures 1 and 2). To maintain chuck accuracy, the 2nd jaw must always be inserted in the same slot even when the jaws are reversed. This slot is identified by a punch mark next to the slot. Always insert the jaws in the order and location shown on the drawings. Turn the scroll counter-clockwise when viewed from the face of the chuck until the outside start of the scroll thread is just ready to pass the slot for the 1st jaw. Slide the 1st jaw as far as possible into the slot. Turn the scroll until the 1st jaw is engaged.

Due to the close tolerances between the slot and jaw, the most difficult part of replacing the jaws is engaging the scroll thread and 1st jaw tooth without binding. Therefore, never use force when replacing the jaws, and if binding occurs, back up the scroll slightly and wiggle the jaw until it is free to move in the slot. Advance the scroll and repeat for the 2nd and 3rd jaws. The scroll thread must engage the first tooth in the 1st, 2nd and 3rd jaws in order.

A set of replacement jaws, P/N 1141 is available. Should it become necessary, please return your chuck to the factory so that we may replace the jaws and check the alignment before returning it to you. In the case of a damaged chuck body, replacement of the entire chuck is usually more economical than attempting repairs.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18



WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

UNITurn & UNIMill
The cool tool

4-JAW CHUCK
P/N 1030 (3.125") and P/N 1044 (2.5")

Because of the varied uses of the 4-Jaw Chuck it would be impossible to write a comprehensive set of safety rules to cover every specific use, other than simply suggesting the use of liberal amounts of "Common Sense". If you're not sure of your set-up, it probably isn't good enough. Get a machinist with more experience to advise on a safe set-up.

Be sure to remove the chuck key before turning the spindle on. Work Safely!

The screws that move the jaws are 20 threads per inch (T.P.I.). A complete revolution is .050". If you keep this number in mind when indicating a part in, it can speed up the process.

First, use the lines machined on the face of the chuck to roughly align the part concentric with the chuck. With an indicator, read the run-out. Move the jaw closest to the high or low point 30% of the total indicator reading in the proper direction.

NOTE: We recommend the 30% figure because the high point of a part will very seldom line up with a jaw. Moving a jaw too much can cause "chasing your tail", or simply moving the high point around the chuck.

EXAMPLE
The indicator shows a .030" run-out. 30% of .030" is approximately .010". If one revolution of the jaw feed screw is .050", then a little less than a 1/4 turn will be .010". Back the jaw out this amount and tighten the opposite jaw. Do **NOT** tighten the jaws beyond "snug" until the part is running within .005" T.I.R. (Total Indicated Reading). Repeat this process until the part runs within your specifications. Once the part is running within .002" T.I.R. it can usually be "brought in" by a final tightening of the jaws. It should also be noted that the chuck jaws are ground with a slight angle to allow the jaws to apply equal pressure to the tip and base when properly tightened. This angle amounts to less than .001" on the jaw surface.

When reversing jaws, be sure not to force a jaw onto the guide rails with the screw. "Wiggle" the jaw as the screw is advanced until the jaw moves in unison with the screw without binding.

If an off balance part has to be run, be sure to turn the motor on at a low RPM setting and bring the speed up slowly; never go past the point that the machine starts to vibrate.

JAW OPENING RANGES
The 3.125" 4 - Jaw Chuck (P/N 1030) opens from 3/32" (2 mm) to 1-1/2" (38 mm) in standard position and up to 2-3/4" (70 mm) with the jaws reversed. The 2.5" 4-Jaw Chuck (P/N 1044) opens from 3/32" to 1-3/16" (30 mm) standard and to 2-1/4" (56 mm) with the jaws reversed. Both chucks have a .687" (17 mm) through hole with a 3/4"-16 thread.

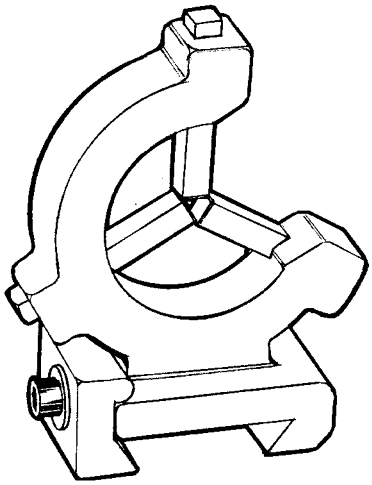
REPLACING WORN OR DAMAGED JAWS
Should the chuck jaws ever become worn or damaged, we recommend you return your chuck to the factory where we will replace the jaws and assure that the chuck is adjusted within tollerances. If the chuck body is damaged, replacement of the entire chuck is usually more economical than attempting to repair the body. If you wish to attempt the replacement of a jaw or jaws yourself, measure the width of the jaw you're replacing carefully with a micrometer (it is usually .312–.315"), and give us the dimension so we can assure a perfect replacement.

REPLACEMENT PARTS LIST

NO. REQ.	PART NO.	DESCRIPTION
1	1144	Set of 4 Chuck Jaws*
1	1146	4-Jaw Chuck Screw*

* Both P/N 1030 and P/N 1044 chucks use same size jaws and screw.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18



WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

UNITurn & UNIMill
The cool tool

STEADY REST
P/N 1074

All materials have a tendency to deflect away from the cutting tools when you are turning them in a lathe. This tendency is especially noticeable on long, slender parts and long pieces of bar stock. It makes it quite difficult to hold close tolerances. The best way to hold a long part is with a center mounted in the tailstock. However, for one reason or another this is not always possible. As an example, it may be a piece of stock that you want to center drill so that you can mount it between centers, or it may be a part where a center drill hole would ruin the looks of the part. Whatever the reason, steady rests provide a means of supporting the part.

The UNITurn Steady Rest provides three adjustable brass blades mounted in a holder that mounts on the bed of the lathe. These blades can be set to the diameter of the part to provide necessary support while it turns. (For small diameter parts it may be necessary to cut or file off the corners of the blades so they contact the part without touching each other.) Another advantage of the Steady Rest which is often overlooked is the fact that work which is held in position by the rest must turn concentrically with its outside diameter. This means that concentricity is assured when working near the Steady Rest because at that point it must be running perfectly true despite imperfections in the way it is chucked or centered at either end.

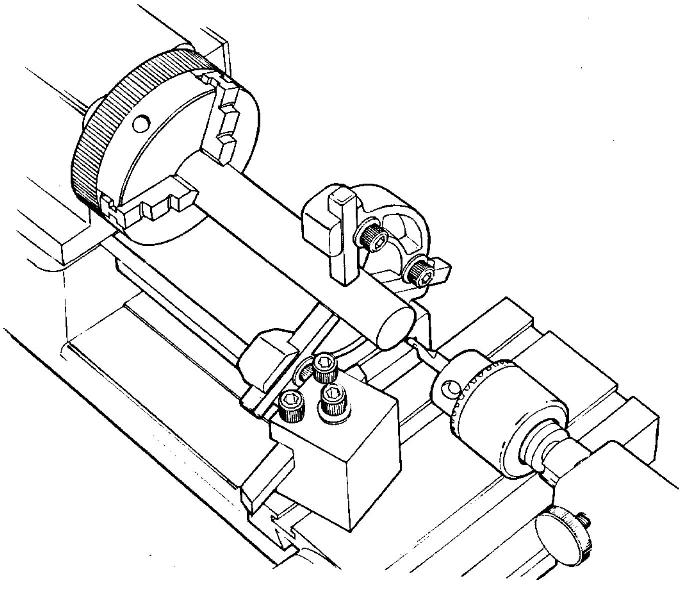


FIGURE 1—To drill a hole in the end of a long shaft, the lathe is set up with a center drill in the drill chuck which is mounted in the tailstock. The Steady Rest keeps the shaft from wobbling and also assures that the hole will be concentric with the outside diameter of the part.

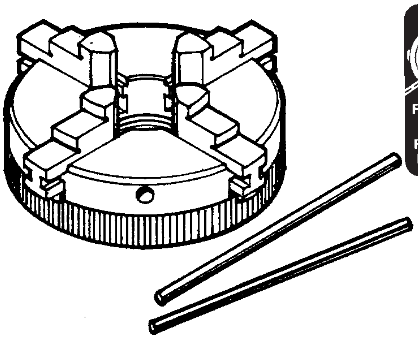
The easiest way to set up a Steady Rest is to first mount the part to be machined in a collet or 3-jaw chuck. Then mount the Steady Rest onto the bed of the lathe and slide it over the free end of the part and up as close to the chuck as it will go. The three blades of the Steady Rest can then be adjusted in until they just contact the part, supporting it but not binding it. Once the blades are set and locked in place, the Steady Rest can be slid back out to support the free end of the part. If you want to check the accuracy of your set-up, you can use a dial indicator mounted on the crosslide. Once you are satisfied with the set-up, apply a drop or two of oil where the blades come in contact with the part, and you are ready to start machining. The UNITurn Steady Rest will accomodate any size part up to 1.75" diameter.

NOTE: A Steady Rest Riser Block (P/N 1290) is now available which makes it possible to use the Steady Rest on the Lathe with the Headstock/Tailstock Riser Blocks in place.

REPLACEMENT PARTS LIST

NO. REQ.	PART NO.	DESCRIPTION
1	1174	Set of 3 Brass Pads
1	1175	Steady Rest Casting
1	1176	Steady Rest Bed Clamp
3	4051	10-32 x 3/8" Skt. Hd. Cap Screws
1	4066	#10 Washer
1	4069	10-32 x 3/4" Skt. Hd. Cap Screw

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18



CAUTION! DO NOT OVERTIGHTEN CHUCK. Use only moderate pressure with the Tommy Bars Supplied.

IMPORTANT! DO NOT TURN THE LATHE SPINDLE ON UNLESS THE CHUCK IS TIGHTENED. Accelleration of the spindle can cause the scroll to open the chuck jaws if they are not tightened!

UNITurn & UNIMill
The cool tool

4-JAW SELF CENTERING CHUCK
P/N 1075

Self centering chucks are designed to have all the jaws move in unison. The jaws are driven by a spiral scroll when the knurled ring is turned. Self centering chucks will never duplicate the accuracy that can be attained with jaws that are moved independently, but they will usually "get the job done", saving a machinist much time and effort.

The main purpose of a 4-Jaw Self Centering Chuck is to hold square stock. It can also be useful in holding thin wall round tubing that will collapse easily. Round stock that is held in this chuck must be perfectly round and can not be at all elliptical or one of the jaws will not grip. The same is true for square stock; it must be very square and not at all rectangular to achieve a proper grip with all four jaws.

This chuck is designed so that the jaws can be removed and reversed to hold larger stock. In the normal position, stock from 3/32" (2.0 mm) to 1-3/16" (30.16 mm) can be secured. With the jaws reversed, material up to 2-1/4" (56.0 mm) can be held. The hole through the center of the chuck is .687" (17.46 mm).

REMOVING THE JAWS

When seen from the front, turning the scroll clockwise backs the jaws out. Turn clockwise until all jaws can be removed. The jaws can be identified by the location of the teeth as noted in Figures 1 and 2.

REVERSING THE JAWS

When reversing the jaws, jaws 4 and 2 will go back into the same slots from which they were removed. Jaws 3 and 1 will exchange positions. The order of installation to reverse the jaws is 4-3-2-1. (See Fig. 1.)

To install the jaws in the reversed position, turn the scroll counterclockwise (viewed from the top) until the outside tip of the spiral scroll thread is just ready to pass the slot for the first jaw to be inserted (jaw #4). Slide jaw 4 as far as possible into the slot. Turn the scroll until the jaw is engaged.

Due to close tolerances between the slot and jaw, the most difficult part of replacing the jaws is engaging the scroll thread and first tooth of each jaw without binding. Never use force when replacing the jaws, and if binding occurs, simply back up the scroll slightly and wiggle the jaw until it is free to move in the slot. Advance the scroll counterclockwise and engage jaw #3 next in the slot that previously held jaw #1 (the slot marked with a punch). continue to engage jaws 2 and finally 1.

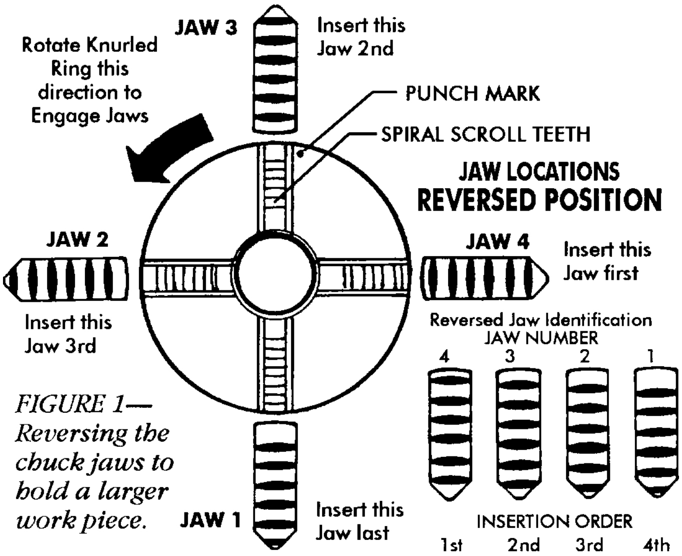


FIGURE 1— Reversing the chuck jaws to hold a larger work piece.

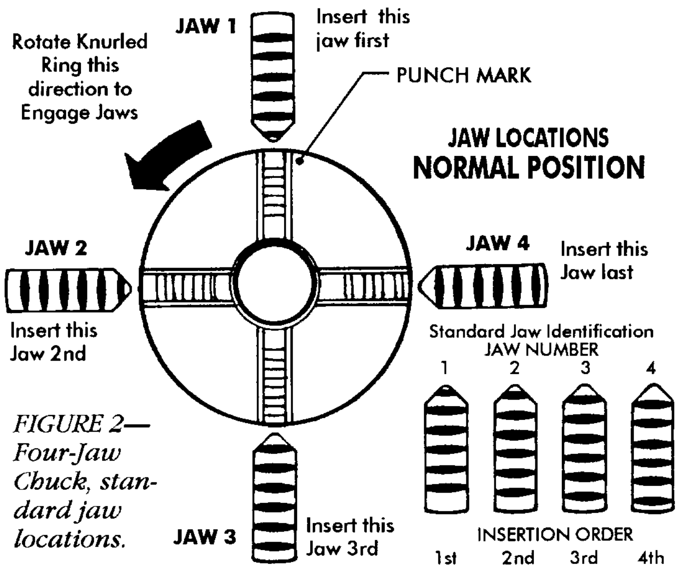
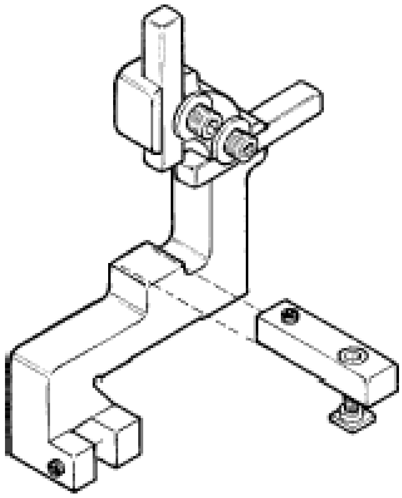


FIGURE 2— Four-Jaw Chuck, standard jaw locations.

When reinstalling jaws in normal position, the order of insertion reverts to 1-2-3-4. (See Fig. 2.)
A set of replacement jaws is available as P/N 1177. Should repair become necessary, please return your chuck to the factory so that we may replace the jaws and check the alignment of the chuck before returning it to you. In the case of a damaged chuck body, replacement of the entire chuck is usually more economical than attempting repairs.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18



UNITurn & UNIMill
The cool tool

FOLLOWER REST
P/N 1090

Purpose of a Follower Rest

The reason this tool is called a “follower” is because the brass supports actually move along with or “follow” the cutter. It is used to support a piece of round stock while it is still being machined to keep the part from deflecting away from the tool. In a normal setup, the UNITurn Follower Rest will lead the tool. (See Figures 1 and 4.)

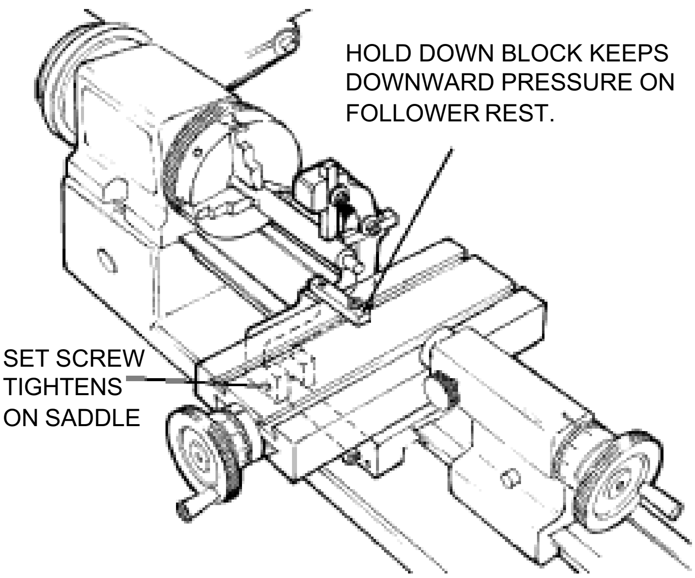


FIGURE 1-Follower Rest installed on lathe. (Tool post removed for clarity.)

A Follower Rest works because it counters the two main forces applied by the tool. When a tool is cutting, the stock wants to climb up on the tool as well as be pushed away. The top brass pad will keep the stock from climbing up, and the brass pad in the rear will keep the stock from being pushed away. The stock will then be cut concentric with the outside diameter because that is where it is supported. It isn’t necessary to have the free end of the stock supported by a center when using a follower, but it does make for a better setup, especially for larger diameters.

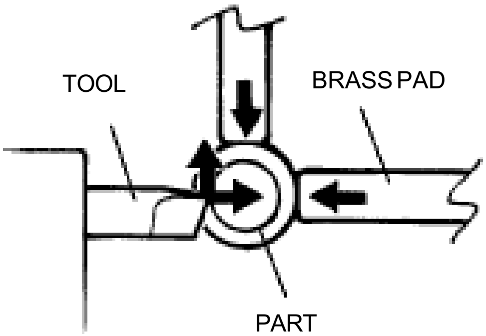


FIGURE 2-Cutting forces on a part and how they are countered by the Follower Rest supports.

When using a center to support the free end, newer UNITurn lathes manufactured after mid-1996 have a cutout in the tailstock to allow it to overlap the table. Older machines may require the use of a tailstock spindle extension (P/N 1220) for clearance. If you are using a tailstock center, the pads should be set by moving the Rest as close to the tailstock as possible, tightening and returning to cutting position.

Mounting the Follower Rest to the Saddle

The UNITurn Follower Rest attaches to the lathe saddle with a flat ended set screw. Push down on the Follower Rest as you tighten this screw so it is clamped flat on the bed. The small block which mounts by means of the crosslide “T” slot is positioned so that the nylon tip set screw pushes down on the machined top surface of the body of the Follower Rest.

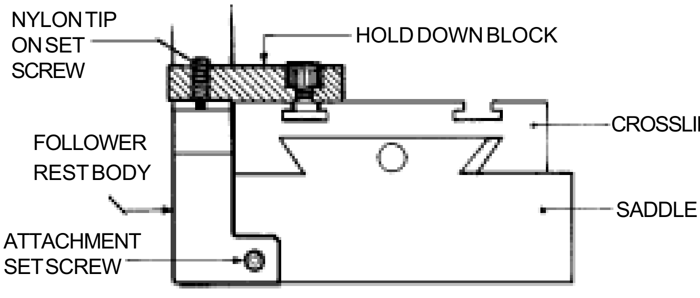


FIGURE 3-Follower hold down block in position.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

This screw holds down the follower rest to counter lifting forces, and its nylon face can also slide on the flat surface so the crosslide can be moved.

In actual use, the tool post should be positioned so only a very small adjustment of the crosslide is required to get the part to finished size.

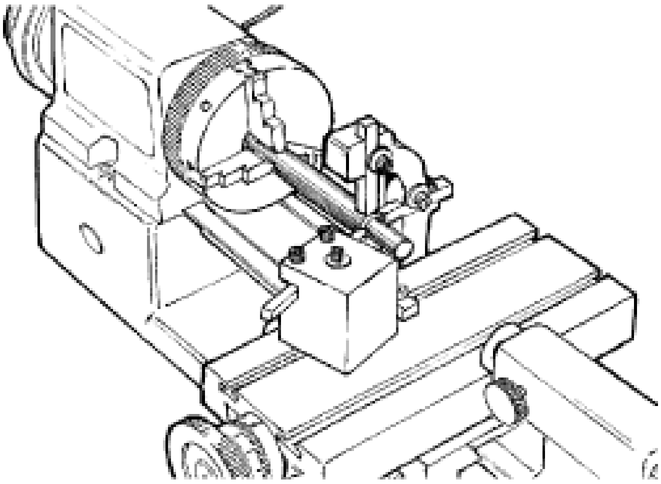


FIGURE 4—Follower rest set up in normal position with pads leading the cutting tool.

SETTING THE POSITION OF THE SUPPORT PADS

To set the pad position, put the round piece you plan to machine in the collet or chuck you will be using. Turn the spindle by hand to make sure the part runs reasonably true. Move the saddle (with the follower rest attached) close to the spindle. Loosen the pad clamping screws, bring the brass pads in contact with the part and retighten the screws to lock them in place. Then move the follower rest back to the position required for the cut and the pads will be aligned with the headstock end of the stock. If you are dealing with very small diameter part, it may be necessary to modify the pad to assure contact. (See Figure 5.)

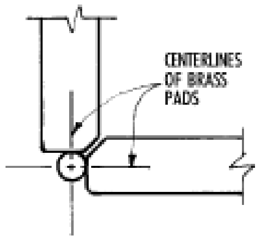


FIGURE 5—Remove the corners of the pad tips to allow them to come closer together for small parts.

With a small diameter rod held in the a chuck or collet, transfer the center of the part to the side of each pad using a scribe. Because of "tolerance buildup", the line may not fall on the exact center of the pad, but that will not effect the function of the follower.

TIPS FOR USING THE FOLLOWER REST

The round stock you use with this attachment should be very round and have a good finish. If the stock is not round, the finished part will have the same shape because the part rotates supported by its outside diameter. A poor surface finish on the part will cause excessive wear on the pads. This in turn can cause your part to taper. To minimize wear, always lubricate the pads with oil when cutting. It would be wise to set up with a piece of scrap of the same material and diameter as your actual stock.

When using a follower of this type, you will usually cut to the finished diameter in one pass. If you need a close tolerance part, it may be easier to turn it slightly oversize, bring it to size with a good, flat mill file and polish it with 320 grit wet/dry paper. If you have a lot of pieces to make, it pays to spend a little extra time getting the setup just right.

MAKING THE CUT

Run the follower rest down the part until the pads are near the end and the tool is just off the end. Dial in the desired depth of cut. If the end of the part is not supported by a center, the part may tend to spring away from the pads a little when not being pressed on by the tool. If the part isn't running perfectly true, it could cause a problem at the start of the cut because the part isn't in constant contact with the brass pads. If this is the case, slip a loop of paper around the part and pull back lightly until the part rests against the pads.

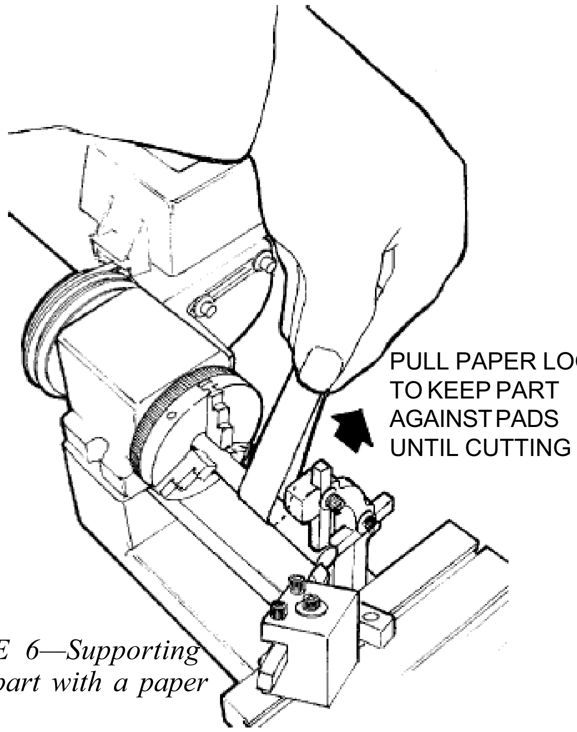


FIGURE 6—Supporting a long part with a paper loop.

Now run the lathe at about 200 RPM and keep the part in position with the paper loop until you begin cutting. If you don't do this, it could cause a problem if the cutter starts to cut and the end of the stock is bouncing around because it isn't running straight or is bent. Take a heavy enough cut to keep the stock firmly against the pads but still larger than the final dimension. Cut about 1/8" (4mm) of length, stop and measure the amount of error and then adjust the crosslide accordingly. (The tool will move but the follower will not.) If the diameter is correct, cut the distance required. By cutting only 1/8" the pads are still supporting the part if you take another cut.

The part size may vary as the pads seat in. Remember to keep them oiled. Keep the RPM down and the feed rate up. A slight radius on the tool tip will improve the finish. When you stop cutting you may have to hold the stock against the pads to prevent "undercutting" as pressure from the tool is released.

TURNING STOCK OTHER THAN ROUND

If you need to turn a round end on material that isn't round (like hex or square stock), the tool must lead the pads so that the pads are running on the round surface cut by the tool. The tool can be mounted almost parallel with the bed to accomplish this. (See Figure 7.) Take your initial (starting) cut with the end of the part held close to the chuck for support. Then move it out into position where the follower rest pads are supporting the newly machined round surface of the part and cut to size. I have found it best to always start with a piece of scrap material identical to your final part for experimentation with the setup. This accessory is not hard to use, but you really need to turn a practice part first to get your speed and feed rates correct.

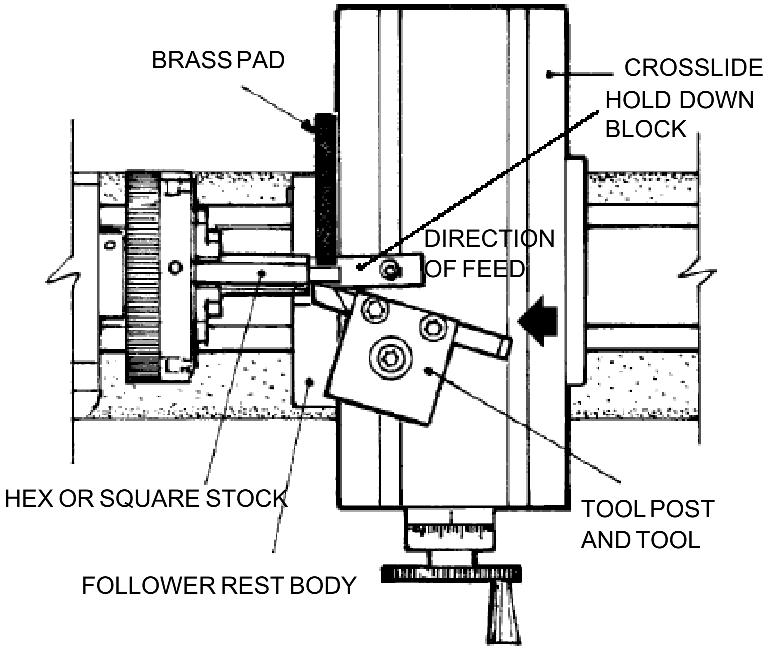
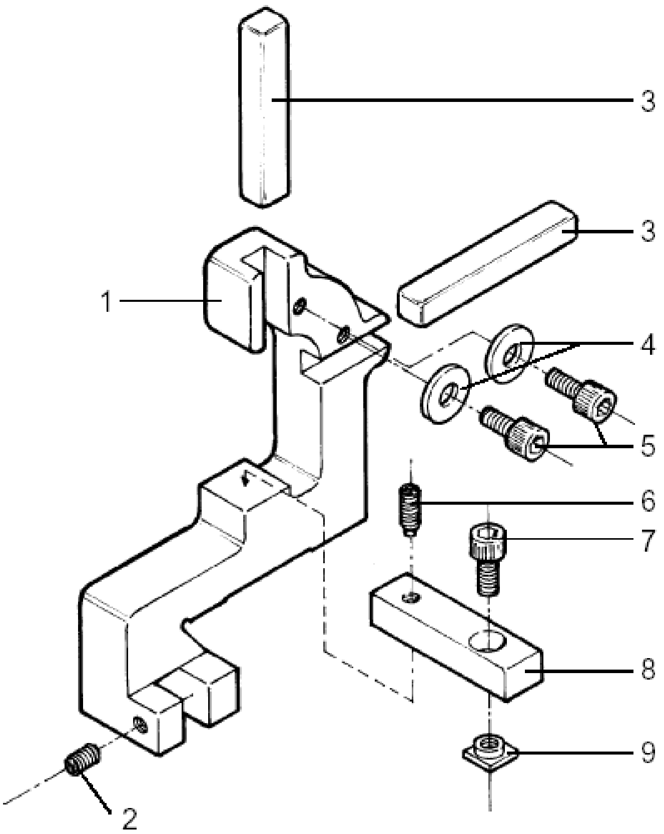


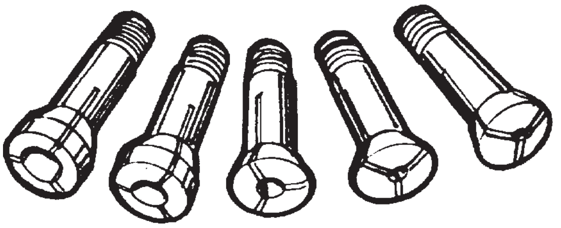
FIGURE 7—Cutting non-round stock.

EXPLODED VIEW PARTS DIAGRAM



FOLLOWER REST PARTS LIST

REF. NO.	PART NO.	DESCRIPTION
1	1087	Follower rest body
2	4060	Flat Point set screw, 10-32 x 1/4"
3	1088	Brass pad (2)
4	4066	#10 Flat washer (2)
5	4051	Skt. hd. cap screw, 10-32 x 3/8" (2)
6	1094	Set Screw w/ nylon head, 10-32 x 3/8"
7	4077	Skt. hd. cap screw, 10-32 x 5/16"
8	1089	Follower hold down block
9	3056	Tee nut, 10-32 (Replacements available as set of 4)



UNITurn & UNIMill
The cool tool

WW COLLETS AND SETS

P/N 1160

Collets provide a quick, easy method of mounting cylindrical parts or bar stock in a lathe with a great deal of centering accuracy. A drawbar which passes through the headstock and threads into the back side of the collet is used to draw the collet tightly into the appropriate adapter. (See Figure 1) The adapter causes the jaws of the collet to close down, gripping the part to be machined. UNITurn Collet Adapter and drawbar (P/N 1161) holds collets with a shaft diameter of .312" to .313". Since many collets are available with shafts of .315" (8 mm) diameter, UNITurn now also offers an adapter for that size as well. (P/N 1156)

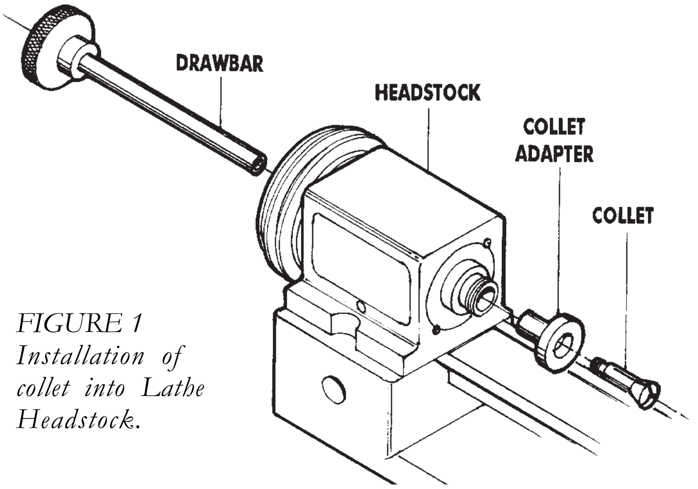


FIGURE 1 Installation of collet into Lathe Headstock.

WW Collets differ from Milling Collets (P/N 3060) in that WW Collets have a hole completely through the collet and drawbar. This is so long material can be passed through the headstock and the appropriate portion machined. The maximum diameter material that can pass through the WW Collet is 3/16" for American size collets and 4.5 mm for metric size collets. WW Collets in larger sizes are sometimes refered to as "Pot Collets". (See Figure 5.)

Collet accuracy may be improved by taking a light cut across the entrance angle of the Collet Adapter with the headstock set at 20° using a boring tool as shown in figure 2. (Refer also to the instruction manual on Taper Turning and Boring.) In most cases, however, collets are accurate enough and do not require this truing operation.

Note also that the collets available from UNITurn are accurate yet economically priced. Should extreme accuracy beyond the tolerances of these collets be required, even more accurate collets are available from other sources and cost not too much.

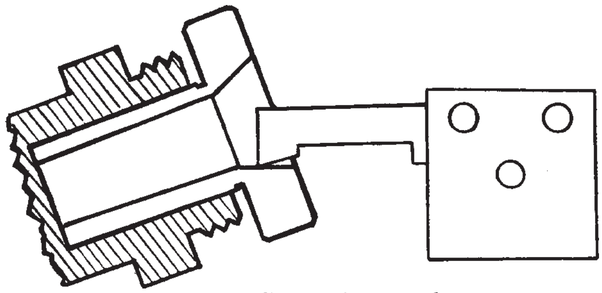


FIGURE 2-- Machining a precise angle on a Collet Adapter

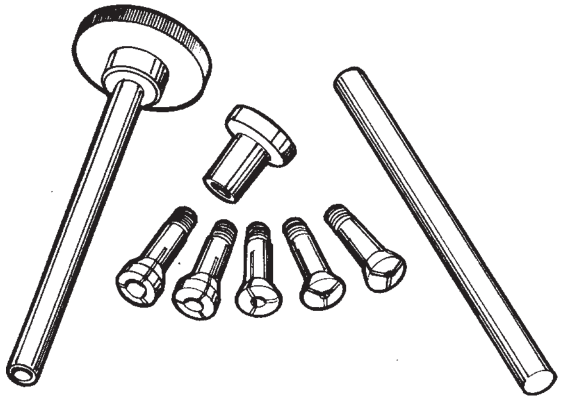


FIGURE 3-- Collet Set with Drawbar, Adapter and Knock-out Bar (P/N 1160, American-- P/N 1178, Metric).

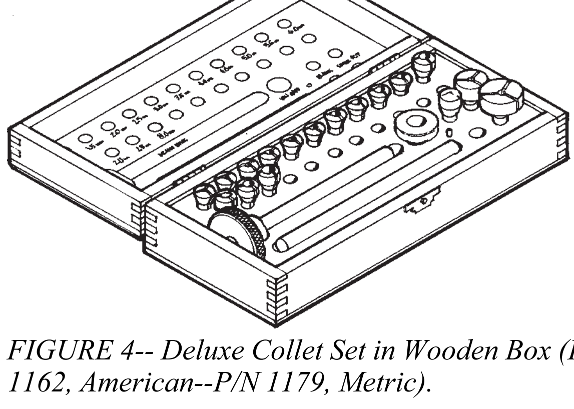


FIGURE 4-- Deluxe Collet Set in Wooden Box (P/N 1162, American--P/N 1179, Metric).

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

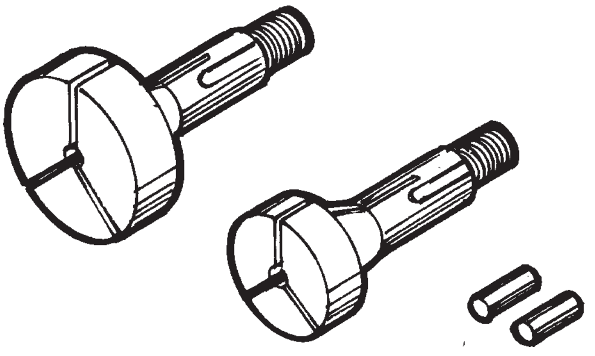


FIGURE 5-- "Pot" or "Step" Collets and Dowel Pins (P/N 2101, 1" and P/N 2100, 3/4").

POT OR STEP COLLETS

These collets are designed to hold larger and odd shaped pieces. The collets are split and have a 1/8" hole through. It is your job to bore them to fit your application. This is accomplished by tightening the collet in the lathe on the 1/8" pin supplied and boring the collet to the size needed. The depth of the bore shouldn't exceed .200" (5 mm). The diameter shouldn't exceed .625" (16 mm) on the 3/4" and .875" (22 mm) on the 1" Pot Collets.

NOTE: Pot Collets are designed to hold material only on the face end, not through the collet.

WW COLLETS--AVAILABLE SIZES

INCH

PART NO.	FRACTION	DECIMAL
2051*	1/16"	.063"
2052	5/64	.078
2053	3/32	.094
2054	7/64	.109
2055*	1/8	.125
2056	9/64	.141
2057	5/32	.156
2058	11/64	.172
2059*	3/16	.188
2060	13/64	.203
2061	7/32	.219
2062	15/64	.234
2063*	1/4	.250
2064	17/64	.266
2065	9/32	.281
2066	19/64	.297
2067*	5/16	.313

* Indicates Collets included in set P/N 1160

METRIC

PART NO.	MM SIZE	DECIMAL
2068	1.5 mm	.059"
2069*	2.0	.079
2070	2.5	.098
2071*	3.0	.118
2072	3.5	.138
2073*	4.0	.158
2074	4.5	.177
2075*	5.0	.197
2076	5.5	.217
2077*	6.0	.236
2078	6.5	.256
2079	7.0	.276
2080	7.5	.295
2081	8.0	.315

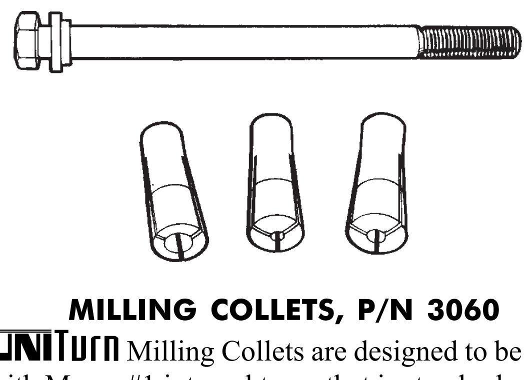
* Indicates Collets included in set P/N 1178

NOTE: Special Collets can be ordered within the range of .050" to .320". When ordering special sizes, please use Part Number 2082 followed by the desired size in inches. (Example: P/N 2082-.193")

PLEASE ALLOW AT LEAST 5 WEEKS DELIVERY FOR SPECIAL ORDER COLLETS.

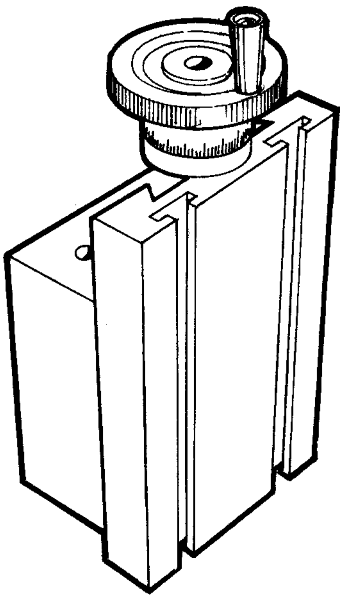
Collet Blanks (P/N 2050) are available. These can be machined to any custom size you desire for special projects.

You may also order the Wooden Box and Insert only from the Deluxe set to create your own custom set. Order Part Number 1170.



MILLING COLLETS, P/N 3060

UNITURN Milling Collets are designed to be used with Morse #1 internal taper that is standard on the spindle of both the UNITURN lathe and mill. Because of the shallow angle of the Morse #1 taper when the drawbolt is tightened, greater clamping force can be applied when compared to the clamping pressure of WW Collets; therefore, we recommend the use of these milling collets for holding miniature size end mills, #1 and smaller center drills (1/8" shank), and assorted other small cutters.



WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

UNITurn & UNIMill
The cool tool

VERTICAL MILLING TABLE

P/N 1185 (Inch)
P/N 1184 (Metric)

Whether you're milling with the Vertical Milling Column or the Vertical Milling Table, some of the same basic rules apply. Here is a brief summary of those rules:

1. This is a small, light duty mill and shouldn't be used to remove vast amounts of unnecessary stock that could be easily removed with a hacksaw. Get stock as close to size as possible before starting.
2. Loads involved for milling are a lot higher than lathe turning. Vibration level is also a lot higher; therefore, more attention must be paid to gib adjustments. They should be kept snug, but not overtightened.
3. End mills must run true and must be sharp. Holding end mills in a drill chuck is a poor method. Milling collets should be used for this purpose. When cutting aluminum, run the motor at top speed and take light cuts.
4. Fly cutting is an excellent way of cutting stock from flat surfaces.
5. Learn to use a dial indicator.
6. Shims may be required to properly align the machine. Normally, standard machine alignment will be good enough for most work unless it is exceptionally large or has to be extremely accurate.
7. A good vise is a must.
8. Often more time will be spent making fixtures to hold work than doing the actual work. There aren't any short cuts in this type of work. If your part comes loose while it is being machined and is destroyed, more time is lost than that saved in a quick set-up.
9. Always try to have one point to measure from. Don't machine this point off half way through the job and leave yourself with no way of measuring the next operation. **PLAN AHEAD!**
10. A good rule for machining operations is, if the tool chatters, reduce speed and increase feed.

It takes a long time to accumulate the knowledge, tools and fixtures to do the tremendous amount of different types of operations involved in milling. Don't get discouraged by starting a job that is too complex.

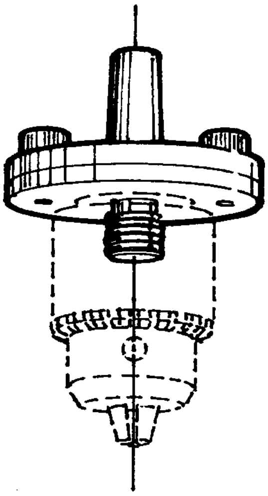
Refer to UNITurn INSTRUCTION GUIDE (P/N 5326) for Milling Setup and Operations.

VERTICAL MILLING TABLE PARTS LIST

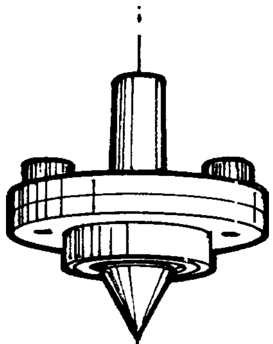
NO. REQ.	PART NO.	DESCRIPTION
1	1183	Milling Table Base
1	4005	Handwheel, Inch (P/N 4105, Metric)
1	4021	Slide Screw, Inch (P/N 4121, Metric)
2	4025	Tee Nut
3	4052	Cone Point Set Screw, 10-32 x 3/16"
2	4073	Skt Hd Cap Screw, 10-32 x 2"
1	4082	Gib Lock
1	4088	Crosslide
1	4089	Slide Screw Insert, Inch (P/N 4189, Met.)
1	4098	Crosslide Gib

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone: +43-2236-892 666 fax: +43-2236-892666-18

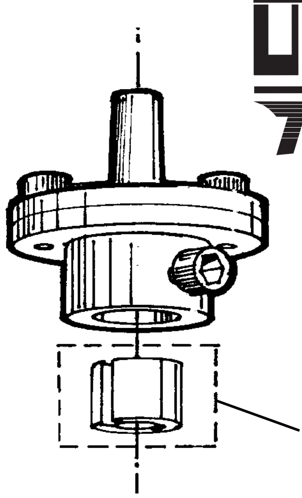
UNITurn & UNIMill
The cool tool



ADJUSTABLE
CHUCK HOLDER
P/N 1202
(Drill Chuck NOT
included.)



ADJUSTABLE
LIVE CENTER
P/N 1201



ADJUSTABLE CUSTOM
TOOL HOLDER
P/N 1203
Your own custom made
split collet 5/8" O.D.
(Not Included)

ADJUSTABLE TAILSTOCK TOOL HOLDERS
P/N 1201, 1202, 1203

The UNITurn lathe has come a long way since its original conception 25 years ago. It started out as a machine that could be manufactured and sold at a very reasonable price, but the accuracy was such that it had limited use.

When the company was purchased in 1974 and started to produce these machines, we completely changed the manufacturing methods and "tightened the tolerances". The biggest improvement in the machines came with the advent of CNC machines (computer controlled) which is how the machines have been manufactured for the last ten years.

Along with the improved accuracy came another set of problems; customers are now using UNITurn tools to do work that, until now, could only be done on machines being very expensive.

The weakest point of the UNITurn lathe design is also the best one; that is, the headstock is removable. This allows taper cutting, milling conversions, riser blocks and numerous other set ups to be made that could never be accomplished without this feature. The negative part of this design is, it's impossible to have perfect tailstock to head-stock alignment. Engineering is always a compromise. In manufacturing the adjustable tool holders we are also admitting we don't have perfect alignment which is the reason for this explanation.

Only someone new to the machine trade would talk "perfect" alignment. In the machine business you talk tolerances even if you can't measure an error because now the error has to be assumed from the tolerances of your method of checking. To maximize the use of the UNITurn lathe we are introducing a series of three tool holders. Holders such as these have always been used in setting up Turret Lathes, and Screw machines in the machine trade to make up for the inaccuracies in machine tools or the lack of room for drill chucks, etcetera.

The UNITurn holders have a Morse #0 taper to fit the tailstock and a choice of three tool holders:

P/N 1201...ADJUSTABLE LIVE CENTER
P/N 1202...3/8-24 DRILL CHUCK HOLDER
P/N 1203...5/8" TOOL HOLDER

These holders are simple to use. The holders are divided into 2 parts with flanges. These flanges are bolted together with 2 screws. The clearance holes for these screws allow the front to be adjusted in relation to the rear. The rear section has a witness mark (hole). This hole should always be located at the top so the holder is located the same way in the tailstock.

The accuracy that is attainable is governed by the amount of skilled effort you put forth. Before starting, it's wise to clamp your headstock square with the bed. This can usually be accomplished by loosening the headstock and pushing back evenly against the alignment key (located under the headstock) and retightening.

To line up the tailstock chuck, put a scrap piece in the 3-Jaw that sticks out approximately 3/4" and face and center drill the end with your present Morse #O arbor and drill chuck. The center drill will find center of the stock even though the chuck may not be lined up perfectly.

Next, mount the drill chuck on the adjustable arbor with the center drill still in it. Bring the tailstock up until the center drill is in the just drilled hole with the screws loose. Tighten when you feel it's on center in the hole. Repeat this process to assure alignment using the new adjustable

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

arbor. This should be close enough for a drill chuck because drill chucks are only accurate within .003" when new at best.

Accurate drill chucks cost approximately 4 times as much and only run within .002". They might claim .001", but I haven't seen it unless you have brand new everything. They are not a good investment for the home shop machinist.

With the drill chuck aligned you can use the same set up to align the adjustable live center by putting the point into the center drilled hole and tightening the screws to start.

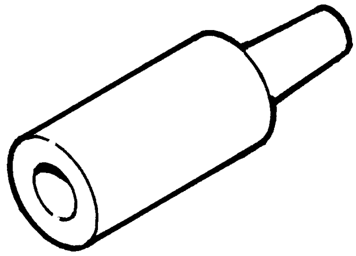
Turn a test bar and correct any error. This can be time consuming and adjustments can be made by never locking the screws so tight that you can't move it with a few taps of a small mallet. When aligned to your satisfaction, screws can then be tightened completely.

The adjustable tool holder allows larger drills and cutting tools that can't be held in our standard drill chuck. Tools are held in a split bushing that can easily be made. The outside diameter has to be .625" and the inside diameter to fit the tool you wish to hold. The bushing is then split almost through with a hacksaw or slitting saw in the direction of the hole. The tool can now be clamped in the holder using this split bushing.

We personally don't believe a person should try and get these any more accurate than you realistically need. Machining is a process that takes place under high loads and temperatures. A perfectly aligned machine doesn't produce a perfect part without the skill of an operator who copes with the many variables. The skill of machining is making parts that are of a closer tolerance than the machine you are working with was built. If you cut a slight taper on a lathe there is nothing wrong with straightening it with a file (flat mill) and polishing with 320A Wet/Dry paper. This should only take a couple of minutes. Trying to align your machine could take hours only to find the machine aligned satisfactory, but your cutter was dull and below center. Please, don't become a machinist that can never get a job done correctly because of the equipment on hand. We've seen beautiful parts produced in machine shops on equipment they wore out 20 years ago; it's the machinist that build these parts not the machines!

PARTS DESCRIPTION

NO. REQ.	PART NO.	DESCRIPTION
1	1204	Adjustable Tool Back
2	1205	8-32 x 3/8" Skt Hd Cap Screw
2	1206	#8 Washers
1	1207	9/64" Hex Key
1	1208	Adjustable Live Center Face
1	1209	Adjustable Chuck Arbor Face
1	1210	Adjustable Tool Holder Face
1	1211	10-32 x 5/16" Skt Hd Cap Screw
1	1092	Live Center Point
1	1093	3/8" Bearing



WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

UNITurn & UNIMill
The cool tool

TAILSTOCK SPINDLE EXTENDER
P/N 1220

A miniature Lathe by its nature and design must fit the most function into the smallest space. This means that to turn an 8" long part on a lathe only 15" long, the tailstock cannot have too much overhang over the table. The Tailstock Spindle Extender provides this overhang while retaining the stability of the Tailstock Spindle by not requiring that it be cranked out to its maximum extension to reach out over the table.

Certain setups are made much easier by using the Tailstock Spindle Extender because it adds 1-1/2" to the reach of the Tailstock Spindle.

By using the extender, a part being held between centers can be turned from end to end without having to move the toolpost from one position to another to keep the Crosslide Saddle from hitting the tailstock. Figure 1 below illustrates how a piece of tubing held in a 3-Jaw Chuck and a Live Center can be turned from end to end with one tool post position.

You will no doubt find other occasions as well where this simple tool pays for itself many times over in convenience.

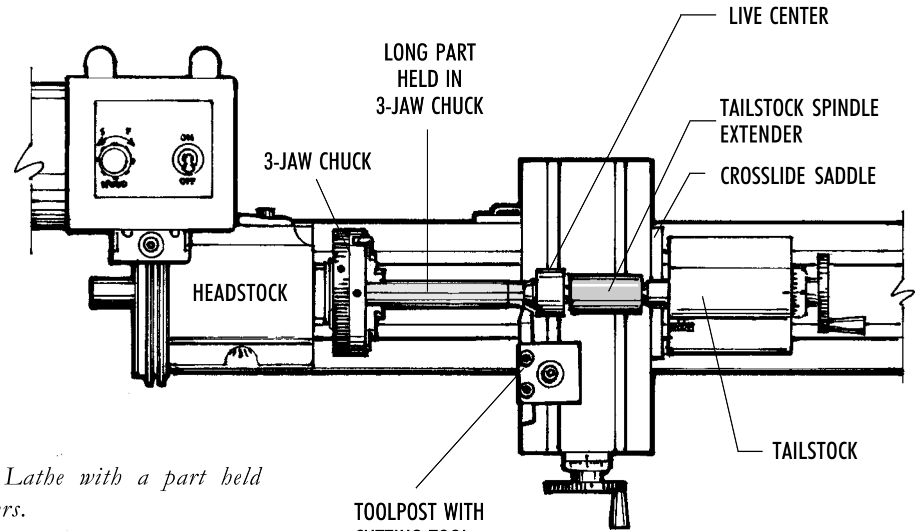
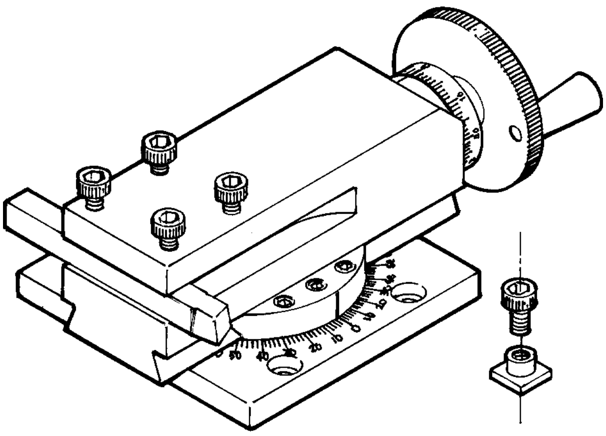


Figure 1
Top view of Lathe with a part held between centers.
The Spindle Extender moves the Live Center out an extra 1.5" so the saddle won't hit the Tailstock when machining from one end of the part to the other in a single setup.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

CAUTION!
Do not overtighten the screws holding the tool or the compound. Distortion of the tool or damage to the crosslide table slots can occur.



UNITurn & UNIMill
The tool tool

COMPOUND SLIDE

P/N 1270 (Metric 1280)

Use of the compound slide

A Compound Slide is used to cut angles or tapers that cannot be cut by "swinging the headstock". (See the instruction manual under the section on taper turning.) The slide has 1-1/2" of movement. The cutting tool can be held on either side or across the end of the compound body.

Actual use of the compound is quite straightforward. Use a properly sharpened tool bit which lines up with the center of the part being cut as there is no adjustment other than shimming to raise or lower the tool. The tool is mounted "upside down" with the cutting tip downward and the compound is used on the "back" side of the part.

Four T-nuts are provided to hold the base of the compound to the table for a very secure mount without overtightening. Make sure the base is mounted square to the table so the laser engraved angle scale will provide accurate readings.

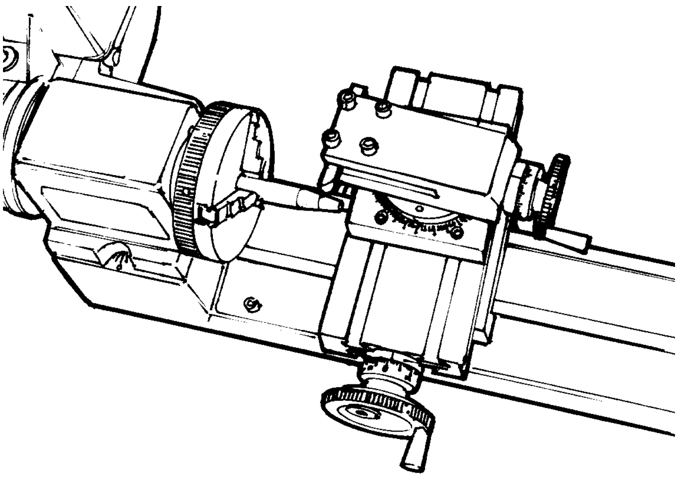


FIGURE 1—Cuting a simple taper with the compound slide eliminates the need to rotate the headstock.

Design considerations in improving the compound

Designing and manufacturing accessories for miniature machine tools often requires a different approach, and the compound slide is a perfect example of this. On a full-size lathe, the compound would normally be mounted on the

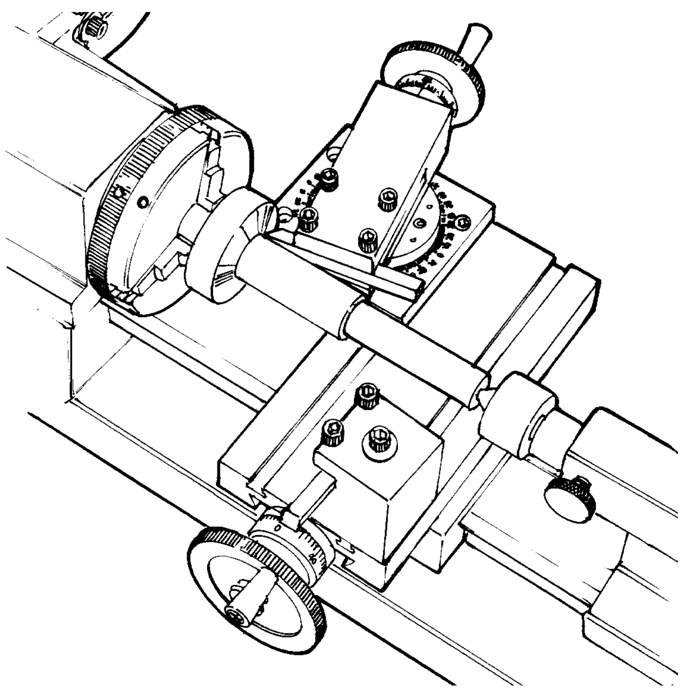


FIGURE 2—The compound can also be offset to allow cuts to be taken close to the chuck. It would be more difficult to hold this piece between centers and cut the taper by offsetting the headstock as might normally be done. Take light cuts when the compound overhangs the table like this.

crosslide and left in place. On a lathe the size of the Sherline, the compound would not only be in the way for many operations, it would add substantially to the initial purchase price of the lathe. Mounting the compound to the front part of the slide limits its movement because of interference with the crosslide handwheel.

Mounting the compound to the rear of the crosslide not only eliminated this interference, it created two additional advantages. First, since cutting work on the "back" side means the surface of the work is moving "up" past the tool rather than down, the tool is mounted upside down with the cutting tip facing downward. This makes the tool less prone to “chatter”, because if the cut gets too heavy, the tool is lifted rather than digging in. Secondly, by mounting the tool only in the upside down position, the slide can be made stronger because it is no longer necessary to leave room for the additional 1/4" spacer required to place the tool at the proper height when using it in the right side up position. This allows the area under the tool to be made thicker.

Adjusting the gib

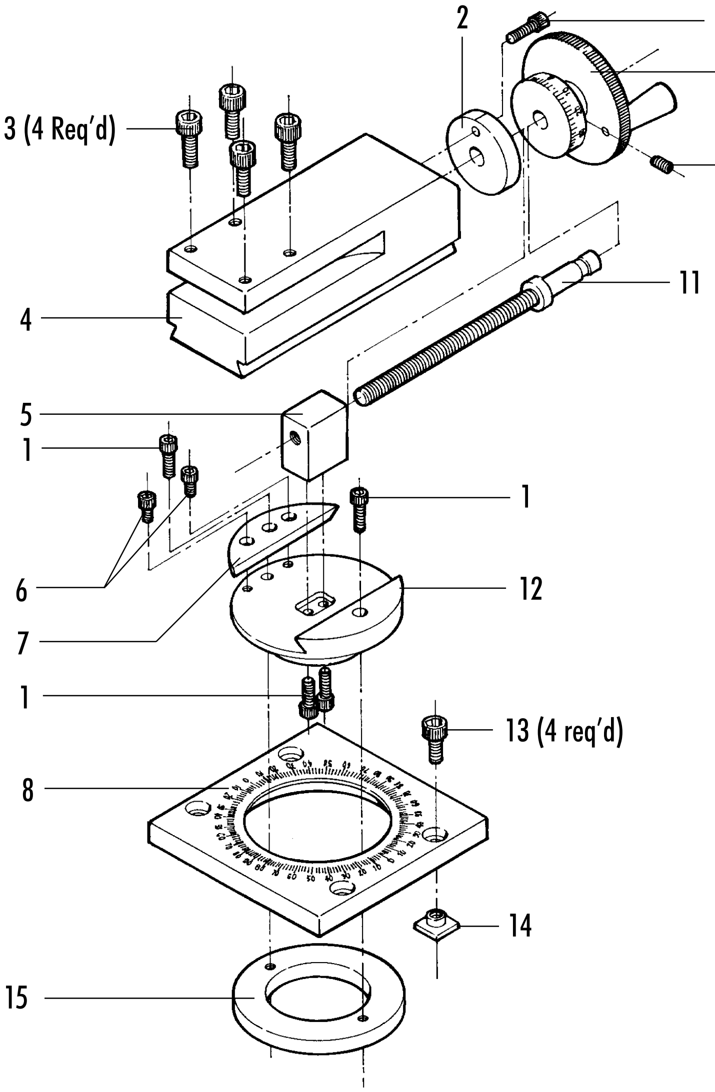
The gib grips the dovetailed base of the compound slide tool post and controls both side-to-side play and freedom of movement. If the gib is too tight, the handwheel will be difficult to turn. If the gib is too loose, the tool post will have excess side-to-side play. To adjust the gib (see reference number 7 in the exploded view below), first loosen all three screws holding it down. (The center screw is used to lock the base in place, but it must be loose before the gib can be adjusted.) With one hand, grip the rotating base and the gib and squeeze the gib firmly against the dovetail of the slide tool post. While still holding it, tighten the screws on either end of the gib. Try the handwheel and see that the slide moves freely. If it is too tight, loosen the two screws and adjust again, this time not squeezing quite as hard on the gib. Clean and lubricate the gib and dovetailed slide with light oil periodically.

Locking the base in position

When the base is rotated to the desired angle, lock it in place by tightening the center screws on the gib and rotating base. Loosen both screws to rotate the base.When locking the base, do not overtighten the screws, as the design is quite efficient and provides a large amount of surface friction area on the clamp ring. Periodically lubricate the clamp ring and base with light oil for smooth rotation.

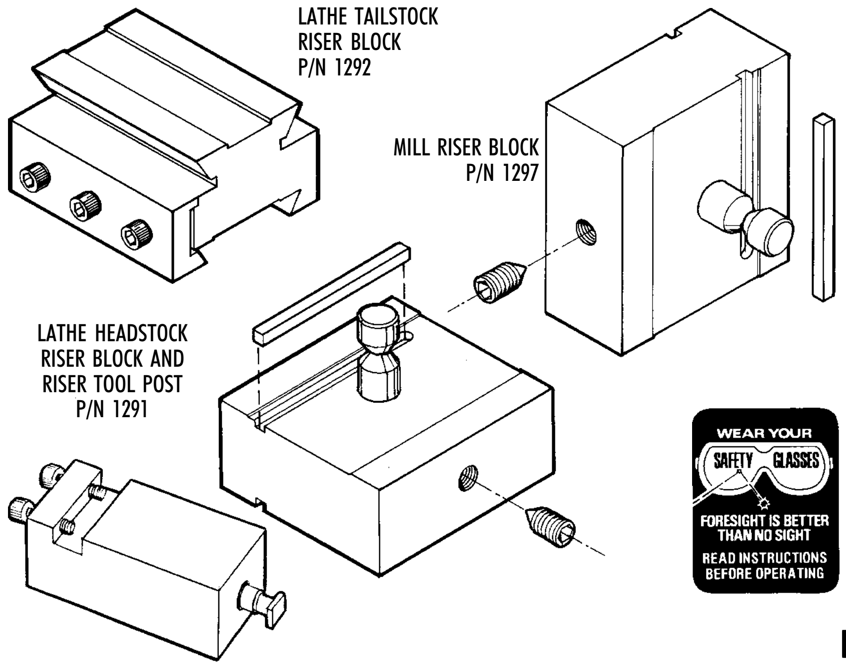
Joe Martin, President and Owner
Sherline Products Inc.

EXPLODED VIEW

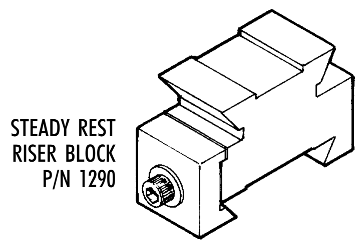


PARTS LIST

REF. NO.	PART NO.	DESCRIPTION
1	4053	5-40 x 3/8" SHCS
2	4028	Thrust collar
3	4067	10-32 x 1/2 SHCS
4	1276	Compound slide tool post
5	1274	Compound Nut
	1264	Compound Nut (Metric)
6	1271	5-40 x 3/16" SHCS
7	1277	Compound gib
8	1272	Compound base
9	4005	Handwheel (Y-axis)
	4105	Handwheel (Y-axis, Metric)
10	4052	Cup pt. set screw, 10-32 x 3/16
11	1278	Compound slide screw
	1268	Compound slide screw (Metric)
12	1273	Compound rotating base
13	1085	10-32 x 1/4" SHCS
14	3056	10-32 T-nut
15	1275	Clamp Ring



UNITurn & UNIMill
The cool tool



RISER BLOCKS

P/N 1290, 1291, 1292, 1297

The purpose of the riser blocks is to extend the capabilities of the UNITurn-Lathe and UNIMill DeLuxe-Vertical Mill. The lathe was never designed to turn metal parts of the diameters that can be accommodated with these accessories; therefore, extreme care must be taken in the form of light cuts and low RPM when turning large diameters.

Another point to be considered is accuracy. When you start clamping several pieces together, alignment will suffer. In the real world of machining, spindles are aligned by indicating, not with pins or keys. This wouldn't be the best way for hobbyists to start, and I believe the methods we use give our average customer machining capabilities they could never have without experience. As your projects get more and more complex, these methods may not be good enough. We manufacture adjustable tool holders (for more information read instructions for P/N 1201, 1202, 1203) to help eliminate some of these problems caused by misalignment. If you believe alignment could be a problem, machine a piece of scrap as a test piece to get the machine lined up. Don't risk a part you may have a lot of work in.

You may have to use a little ingenuity when turning large diameters because of the limited crosslide throw on standard machines. The purpose of the mill riser block (P/N 1297) is to get the spindle farther out from the column. This allows you to work farther in from the edge. There isn't any difference between the lathe and mill riser block except the lathe P/N 1291 comes with a corresponding tool post.

INSTALLATION

Remove the headstock by loosening the screw that holds it onto the lathe or mill and lift it straight off. Now install the riser block using the keyway to align it. Do this by pushing the riser block back towards the keyway without a twisting motion. Put the headstock back with or without the keyway depending on your next machining operation (taper cutting).

It is necessary to remove the handwheel at the end of the bed to remove the tailstock. Install the Tailstock Riser Block. You may have a slight problem fitting this up. It is a very difficult part to make because dovetails can't be measured or machined easily. The biggest problem we have encountered is the "tip" of the dovetail on the lathe bed may interfere with the riser block. A couple of passes with a file (see figure #1) should fix it. Riser Blocks made after 11/93 are of a two-piece design that in most cases eliminates this fitting problem.

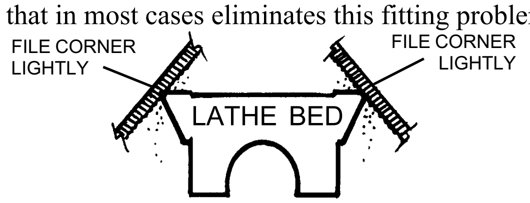


Figure 1—Filing corners of bed dovetail for better fit of Tailstock Riser Block.

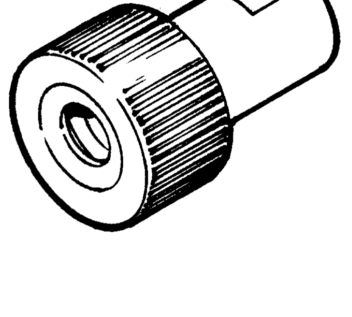
When replacing the handwheel try and let the set screw pick up the same indentation so you don't "chew up" the end of the lead screw shaft.

We at UNITurn, hope you find these to be useful accessories.

REPLACEMENT PARTS LIST

NO. REQ.	PART NO.	DESCRIPTION
1	1293	Tailstock Riser Body
1	1294	Tailstock Riser Clamp
1	1295	Headstock Riser Block Body
1	1296	Spacer Block Tool Post Body
1	1298	1/4-20 x 3/8" Flat Head Machine Screw (1291, 1297)
1	1299	Pivot Pin (1291, 1297)
1	1391	Steady Rest Riser Body
1	1392	Steady Rest Riser Clamp
1	4025	Tee Nut (1291, 1297)
1	4026	Head Key (1291, 1297)
–	4033	10-32 x 5/8" Skt. Hd. Cap Screw (1290–1 req., 1292–3 req.)
1	4054	5/16" -18 x 3/4" Cone Point Set Screw(1291,1297)
1	4066	3/16" #10 Washer (1290, 1291)
2	4069	10-32 x 3/4" Skt. Hd Cap Screws (1291)
1	4073	10-32 x 2" Skt. Hd. Cap Screw (1291)

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18



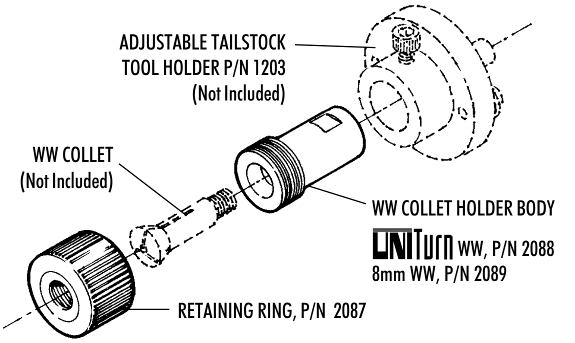
WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

UNITurn & UNIMill
The cool tool

WW COLLET ADAPTER

P/N 2085 (UNITurn WW)
P/N 2086 (8MM WW)

The purpose of the WW Collet Adapter is to allow the use of WW Collets in the Lathe Tailstock. We decided to add this product to the accessory line based on the requests of several watchmakers. They have a need to drill very small holes with the accuracy only a collet can provide. A drill of only a few thousandths of an inch in diameter is easily broken if not perfectly centered. To accomplish this level of accuracy on a UNITurn Lathe, a little extra time must be spent to "perfectly" align the machine. When you consider the alternative is to spend huge sums of money for a Jeweler's Lathe that is far less versatile, we think you will find it is time well spent.



ALIGNING THE HEADSTOCK

The first step is to align the Headstock with the Bed. Loosen the Headstock and push it back evenly against the alignment key (located under the headstock) and retighten. Put a scrap piece of round material about 5/8" (16 mm) diameter by 3" (76 mm) long in the 3-Jaw Chuck, turn the outside diameter to round and face the end with a sharp cutter. Measure the taper that has been cut and eliminate it by moving the headstock with light taps from a mallet and taking another test cut until any error is eliminated.

You now must make the choice of whether you want the headstock to remain removable. A simple way to "lock it in" is to use Locktite™ on the keyway. Make sure the alignment key and keyway are free from oil before aligning the headstock. After alignment, turn the lathe on end and place a few drops of thread locking compound into the keyway. The Headstock can later be loosened by prying with a screwdriver blade in the slot between the bottom of the Headstock and the Lathe Bed when viewed from the Headstock end.

Another method would be to pin the Headstock to the Bed with 1/8" dowel pins after alignment. To do this, carefully remove the Base from the Lathe and drill and ream two 1/8" holes about 2.5" (63 mm) apart through the bed and into the Headstock. You must do this without moving the Headstock on the bed. The Headstock can be removed this way, although it is a little more difficult than using just the keyway. When it is reinstalled, however, it should align closer than with the standard keyway.

(Our personal opinion is that you will give up more than you gain if you lock down the headstock, as most work can be accomplished with standard keyway alignment. However, if the ultimate in accuracy is your goal, the choice is up to you.)

ALIGNING THE ADJUSTABLE TAILSTOCK TOOLHOLDER

The Adjustable Tailstock Tool Holder, Part Number 1203, is designed to allow minute adjustment of the tailstock center. Complete instructions on how to accomplish this adjustment are included with that accessory.

With the Headstock in alignment, a dial indicator can be used to "indicate in" the WW Collet. (See "Use of a Dial Indicator" section in the UNITurn Instruction Guide.) Another method would be to turn a short length on the end of the sample stock down to 1/16". Then chuck up a piece of 1/16" material in a 1/16" WW Collet held in the Adapter. Bring the ends of the two pieces of material right up to each other and, using a magnifying glass, align the ends by eye.

The Collet Adapter comes in two sizes: UNITurn WW and 8mm. The difference between them is .002" diameter on the barrel part of the Collet. The UNITurn WW is .313" diameter and the 8 mm is .315" diameter.

We believe micromachinists and watchmakers will find this accessory to be most useful. By using this simple tool and a little time, you can duplicate the accuracy of a machine costing many times more, yet still retain the great versatility offered by your UNITurn Lathe.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

UNITurn & UNIMill
The cool tool

WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

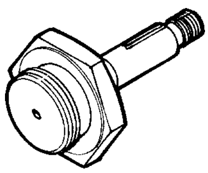
CLOCKMAKER'S ARBORS

Arbors P/N 2090 through 2093
Gearcutting Arbors P/N 2094 through 2096

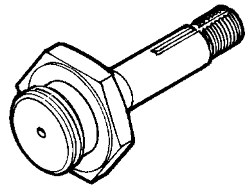
These arbors are not designed to be used on UNITurn lathes, but rather are specifically suited to the types of lathes often used by jewelers and watchmakers. Though highly accurate, these machines are often not very versatile.

The 3/4"-16 Arbors provide a way to mount a UNITurn 3-Jaw or 4-Jaw Chuck to this type of lathe. The 3/8"-24 Arbors allow a 1/4" or 3/8" Drill Chuck to be mounted. The ability to mount these chucks adds a great deal of versatility to this type of special purpose tool. The P/N 2094 Gearcutting Arbor with its #1 Morse taper can be used on your UNITurn Lathe. With use of a P/N 1156 8mm Adapter, either of the 8mm Arbors can also be used on UNITurn lathes, although the normal system for mounting chucks is recommended.

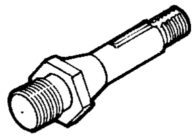
CLOCKMAKER'S ARBORS



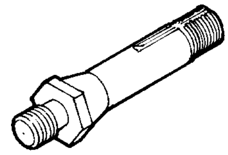
P/N 2090
Clockmaker's Arbor
8mm to 3/4"-16



P/N 2091
Clockmaker's Arbor
10mm "D" to 3/4"-16



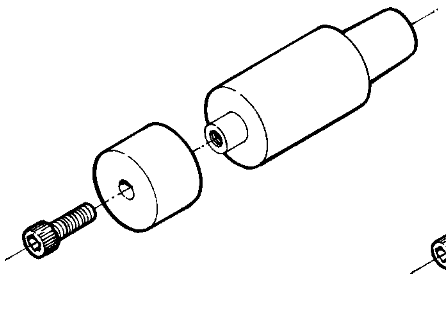
P/N 2092
Clockmaker's Arbor
8mm to 3/8"-24



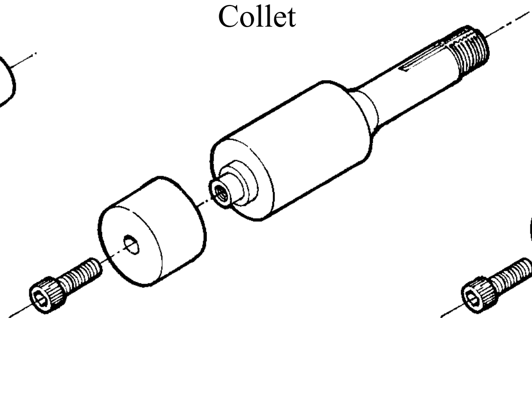
P/N 2093
Clockmaker's Arbor
10mm "D" to 3/8"-24

CLOCKMAKER'S GEARCUTTING ARBORS

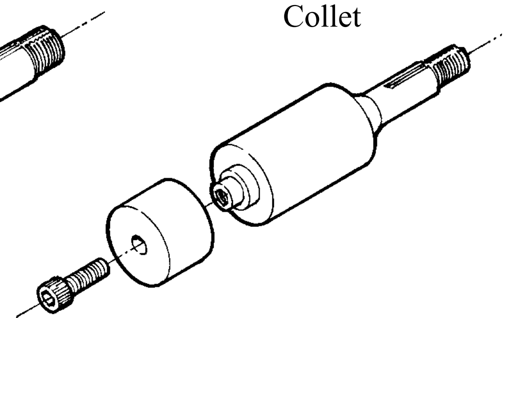
P/N 2094
Clockmaker's
Gearcutting Arbor
7mm to Morse #1 taper



P/N 2095
Clockmaker's
Gearcutting Arbor
7mm to 10mm "D"
Collet

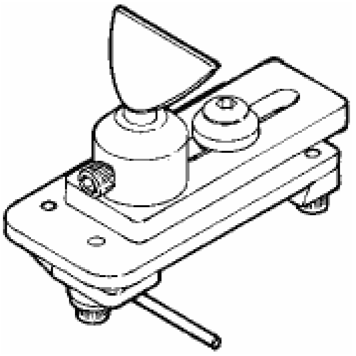


P/N 2096
Clockmaker's
Gearcutting Arbor
7mm to 8mm WW
Collet



The cool tool®
www.thecooltool.com

Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone: +43-2236-892 666 fax: +43-2236-892666-18



UNITurn & UNIMill
The cool tool

W.R. Smith T-Rest

P/N 2110

The Purpose of a T-Rest

This T-Rest was designed by world renowned watch and clock maker William R. Smith. The only change we made was to eliminate locking levers from the post and pedestal base and replace them with 10-32 screws for production reasons. The same 5/32" hex key that comes with your UNITurn lathe can be used to adjust them.

The T-Rest is used to support a metal cutting tool called a "graver" which is hand held rather than held in a toolpost like a conventional lathe tool. This is a traditional method of cutting metal shapes that has long been used by watch and clock makers. It is also used by some instrument makers, modelmakers and machinists.

Because the tool is hand held, there is more "feel" for the cut that is being made. Certain shapes like ball ends, curves and special notches or ridges which would be difficult to make with conventional tools can be done quickly and easily with this technique. It can yield very precise results in the hands of one skilled in this technique; however, a certain amount of practice may be required for a beginner to turn precise parts using this method. For more information on making gravers and how to use them, click here to go an article by William R. Smith on gravers.

Precautions for Hand Turning

Do not use this tool on parts held in a 3-jaw or 4-jaw chuck. A graver which inadvertantly hits a spinning chuck jaw could be dangerous. Hold parts in a collet for hand turning. Because the tool is hand held, it cannot be held as securely as a tool held in a toolpost, so use it with appropriate caution. The cutting angle, sharpness of the tool, position of the tool point and feed rate of the tool are all critical to how it cuts. In a nutshell, when the angles are right, the tool cuts. When they're not, it doesn't. Experiment as you find the best combination and get a feel for the process.

Turning Speeds and Tool Angles

Mr. Smith suggests a turning speed of about 250-500 RPM for turning a small diameter steel shaft. The speeds listed in speed tables for conventional lathe cutting tools do not really apply to cutting with gravers. The basic machining rule does still apply, however and that is:

"If the tool chatters, reduce speed and increase feed."

Rest the tool shank on the T-Rest with the point of the tool on the top side. (See Figure 2.) Slide it along on the bottom pointed edge, holding the tool in one of the grips shown in the graver instruction sheet. The tool can be rotated and pivoted to be used in any number of ways to achieve the type of cut you desire. The tool should be raked downwards at the handle end about 5° to 7° for cutting hard steels. For softer materials like brass, the rake angle can be reduced to near 0° to keep the tool from biting too deeply into the softer metal.

The angle of entry of the tool into the part varies. Start at about the part centerline and move the tool up or down slightly varying the angle until you find a position where it cuts best. You can pivot the tool left and right using pressure from your finger to swing an arc to cut a radius. As the leading edge of the tool bites in, the heel rubs on the part keeping the tool from digging too deeply. Using this method you can achieve very subtle control of your cut.

I suggest you "break" three of the sharp edges of your graver slightly with a stone so they slide smoothly on the top of the T-Rest. If the edges are left sharp they will bite in rather than slide. Mr. Smith gave me a "crash course"

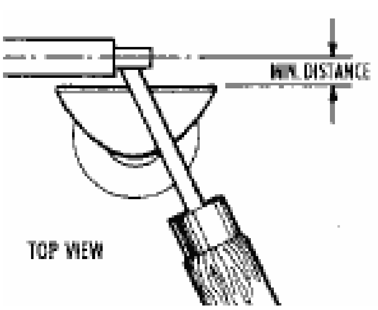


FIGURE 1—The T-Rest is set close up to the work so that the overhang of the tool is minimal. This gives you better leverage on the tool should it dig into the part. (Tool is shown being used on its side.)

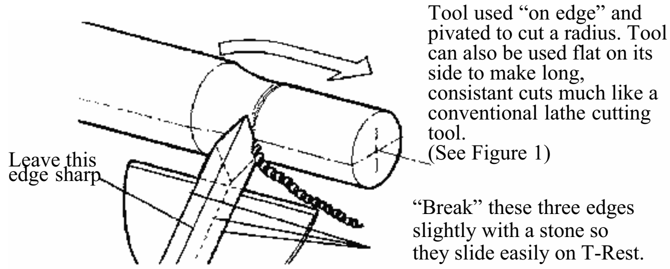
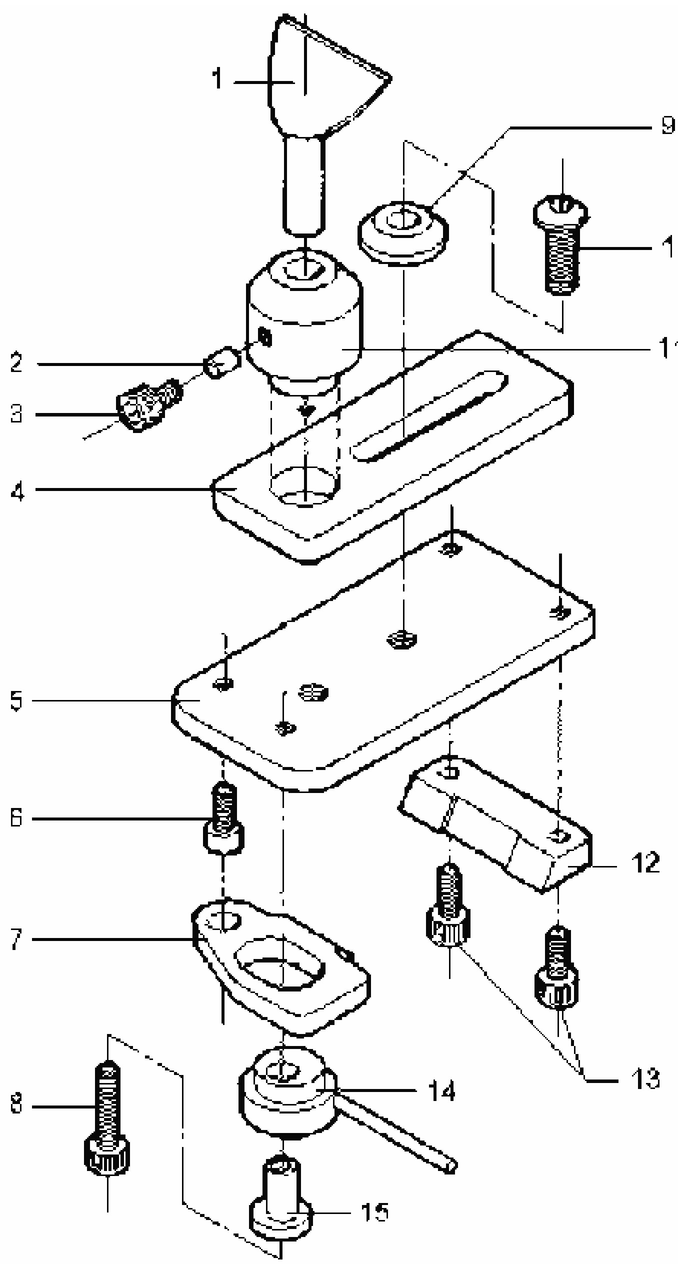


FIGURE 2—Metal peels from the edge of the tool when you find the right cutting angle.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone: +43-2236-892 666 fax: +43-2236-892666-18

EXPLODED VIEW



NOTE: The small brass cylinder (Part Ref.No. 2 in Exploded View above) is inside the hole in the side of the T-Rest Pedestal (Ref.No. 11). It is tightened against the shaft of the T-Rest and its soft material prevents damage to the shaft. If the T-Rest is raised too high in its hole, the brass cylinder can be pushed into the shaft hole, pre-venting the shaft from being pushed back down. If this occurs, remove the T-Rest and use a small screwdriver blade to push the cylinder back into its hole from inside the shaft hole. Then reinsert the T-Rest, adjust to proper height and retighten the locking screw (Ref.No. 3).

on using gravers at his home in Tennesssee, but I still don't feel I have the expertise to write complete instructions for their use. Mr. Smith has kindly given permission to use his instructions on making gravers which are included along with this sheet. He is a superb craftsman and gentleman and we appreciate the opportunity of working with him on this project.

Where to Get More Information

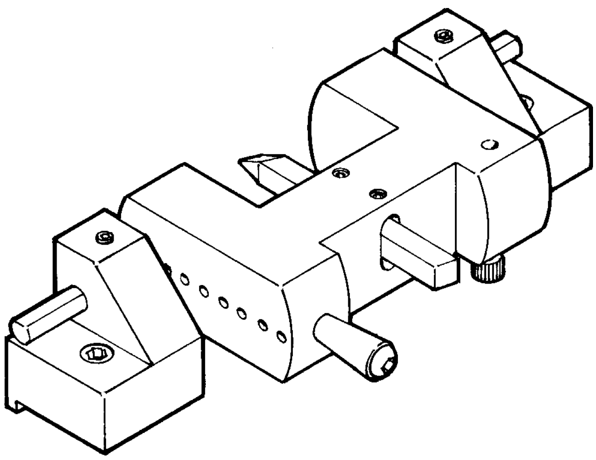
If you are new to the technique of hand turning metal, I suggest you get more information from Mr. Smith or other experts in the horological field. He has published several books and videos that show or describe the techniques required. He may be reached by calling (423) 947-9671. You may write him at: William R. Smith, 7936 Camberley Drive, Powell, Tennessee 37849. His E-mail address is: wrsmith2@aol.com.

If you are going to be making the precise parts required in clocks, models and instruments, you will find that the UNITurn lathe along with the T-Rest will yield results equal to those you would obtain on special jeweler's lathes costing many times more.

REPLACEMENT PARTS LIST

REF NO.	PART NO.	NO. REQ.	DESCRIPTION
1	2120	1	T-Rest
2	2123	1	Brass spacer 5/32” O:D:x 3/16”
3	2127	1	T-Rest Lock Screw 10-32 x 3/16” Skt. Hd Screw
4	2111	1	T-Rest Pedestial Base
5	2115	1	T-Rest Saddle Body
6	2125	1	Cam Pivot Screw(Custom) 10-32 x 5/32” Skt. Hd. Screw
7	2116	1	Cam Follower
8	2126	1	Cam Mounting Screw 10-32 x 11/16” Skt. Hd. Screw
9	2114	1	Hold Down Washer
10	6111	1	Pedestial Hold Dawn Screw 1/4-20 x 5/8” Button Hd Screw
11	2117	1	T-Rest Pedestial
12	2121	1	Dovetail Block
13	4051	2	10-32 x 3/8” Skt Hd Screw
14	2130	1	Cam and Lever Arm
15	2112	1	Cam Spacer Sleeve

WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING



UNITurn & UNIMill
The cool tool

RADIUS CUTTING ATTACHMENT

P/N 2200

Use of the radius cutting attachment

The radius cutting attachment allows you to put an accurate convex or concave shape on a part with a Sherline lathe. Unlike conventional radius cutting attachments that swing the cutting tool parallel to the lathe table, the Sherline tool moves from the center of the part to the top. The idea came from the method toolmakers use to dress (shape) a radius on a grinding wheel with a diamond tool. A ball can also be cut using a two-part process described later in these instructions. Diameters up to 1-1/2" can be cut. A handle is provided that can be used on operations where the tool faces the long end of the “U” shaped cutter body. On operations where the tool faces the other way the handle is not used.

Center height reference numbers

You will notice numbers engraved on the side of each support. These represent the actual distance from the table to the center of the pivot pin for purposes of calculating the center of your radius. The theoretical distance should be .940", but because there is minor variation in that distance due to production tolerances, so each one is measured and the exact distance for that unit is engraved on the side.

Using the stop screw to avoid cutting past center

It is important to note that the cutting edge of the tool is set to cut with the lathe turning in a particular direction depending on which half you are cutting (top or bottom). As the tool passes the centerline the tool cutting edge still faces the same way, but now the work is rotating in the opposite direction. This causes the tool to “drag”. To keep this from happening, a stop screw is provided that will work in most setups. By adjusting the stop screw to hit the table when the radius cutter is at the center position, the tool can be prevented from accidentally dragging on the work. By the same token, always note the direction of rotation and make sure the cutting tool is facing in the proper direction to make your desired cut.

Setting up the tool to cut a convex radius or ball end

The most common application of a radius cutter is to put a radius or ball end on the end of a part. The points of the radius cutter are accurately located on the centerline of the part. (See paragraph above on engraved center height reference numbers.) This makes setting the cutting tool to the proper radius a simple process. Here’s how it is done:

1. Mount the two uprights in the table T-slot closest to the spindle. Pick out the center holes that will give you the best location for the cutting tip of your cutting tool. This is dependent on the final diameter of the ball. The center holes are accurately drilled on .250" centers and only a rough location is needed at this time.

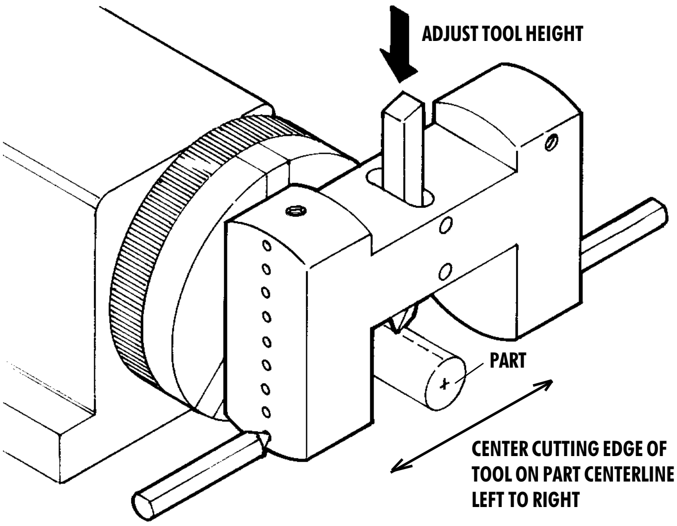


FIGURE 1—Setting the tool depth to cut a full radius on a part already at finished diameter. Adjust the tool until it just touches the top of the part. (Pivot supports not shown.)

2. The quickest way to set a cutting tool is by first turning a finial diameter (twice the radius) on the part you are going to work on with a standard lathe tool before mounting the radius attachment. If the part is already turned to the finished diameter and you wish a full radius on the end, simply raise the yoke to the vertical position and lower the tool until it just touches the top of the part. (See Figure 1.) This will establish the proper radius. It would be safest to set the tool to a slightly larger radius just to be safe and then “sneak up” to the final dimension once most of the material is removed and you can see how close the final cuts are coming to your desired radius. Move the radius cutter away from the end of

the part with the leadscrew. Rotate the tool (assembly) and move the saddle towards the part until the tool is in a position to take a light cut (approximately 0.020") on the top corner. The first series of cuts are accomplished by rotating the tool up and back. Then move the saddle and radius attachment about 0.020" closer. As you get down to the final cuts you will be able to see which way you are off and make final adjustments with the crosslide handwheel to finish up exactly on center to cut the full radius.

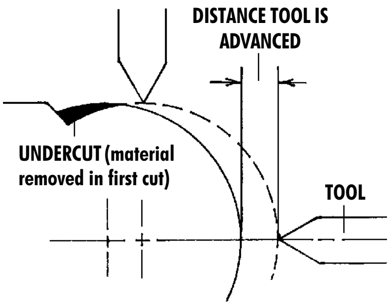


FIGURE 2—The area in black illustrates why you don't want to cut past top dead center until you know you are at final size.

3. Remember that the tool will cut the full amount it has been advanced at the center but will not reduce the diameter at the top and bottom of the part. (See Figure 2 above.) If you move the tool past the top dead center before the tool is cutting at its final position you will undercut the opposite side of the ball. This makes it wise to stay back 10° or so from top dead center until the tool has reached its final position. This creates a "damned if you do and damned if you don't" problem. If you don't cut over the top you can't accurately check the diameter of the ball, and if you do you may scrap out your part. The easiest way out of this situation is to set up on a scrap piece and get the tool set. The radius remains set even if the attachment is removed and replaced on the lathe as long as the tool isn't moved in the radius cutter body.

4. Another way to set the tool would be to accurately cut or mill a gage block to a dimension that is the center height of the part over the table (.940" or the amount engraved on the attachment) **PLUS** the desired radius of the part. Set this gage block on the table and move your tool down to just touch the top of it with the attachment in the vertical position. (See Figure 3.)

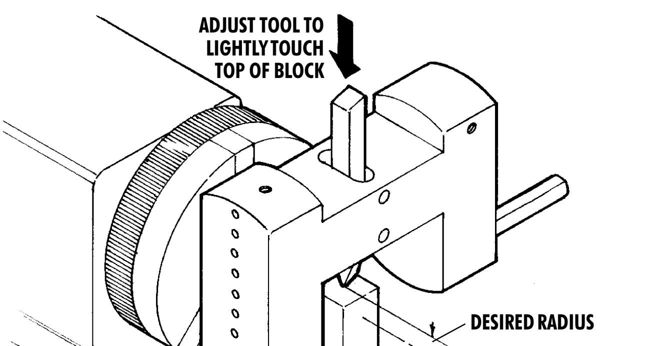


FIGURE 3—Using a gage block to set the tool height for a convex radius. You can use this method when the material has not been turned to the final size of the ball end.

The tool can also be set for cutting a small concave shape using this method only the radius would be **SUBTRACTED** from the 0.940" dimension rather than added for obvious reasons. (See Figure 4.)

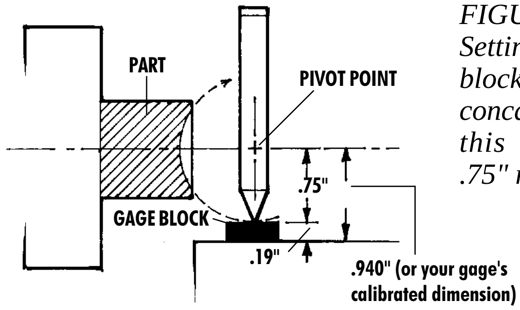


FIGURE 4—Setting up a gage block for a small concave radius. In this example, a .75" radius is cut.

To cut a large concave radius, a height gage can be used to set the tool height. The height gage is set to .940" **PLUS** the desired radius as seen in Figure 5 below.

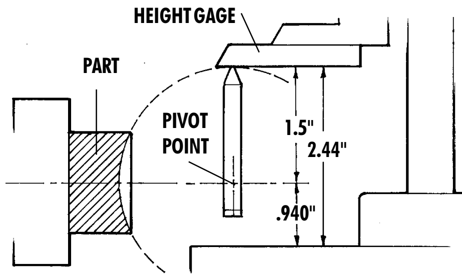


FIGURE 5—Using a height gage to set up a large concave radius. In this example, a 1.5" radius is cut.

Correcting for an incomplete radius cut

When cutting a full ball end, if your first attempt measures undersize, you will have to scrap out the part and start over. That is why I suggest you start with what you know will be a slightly oversize cut. Once you are near the final size, here is how to adjust the cutting tool depth to get the exact size:

1. Measure the diameter of the ball you have cut. You can't reset the tool until you know the diameter it is actually cutting.

2. If, for example, the ball is 0.010" oversize, the tool must be moved in the cutter body 0.005" (half the desired distance) closer to the part. To do this, first record the leadscrew handwheel setting with the backlash taken up in the cclockwise direction and the tool touching the end of the center of the part. This should be done right after your last cut was made so you know the tool is just touching the part.* Now use the leadscrew handwheel to back the saddle/radius cutter assembly up more than the correction needed. Turn the handwheel clockwise again with the difference calculated in. The tool should now be .005" from the part. Loosen the set screws holding the tool in the body and move the tool until it just touches the part*. Tighten the set screws and make your final cut.

*NOTE: Whenever you move a tool up to touch a part to set its position, don't push it into the part. Make sure it barely touches. Pushing a tool into a part will cause it to take an extra couple of thousandths off on the next cut, and your part will come out undersize.

Cutting a concave radius

Full convex radii are easy to measure because you can use a caliper or micrometer. A concave radius is more difficult to measure. It is better to spend the time accurately clamping the tool using a height gage than trying to check your radius with a template you can't view accurately. Some things in machining have to be controlled with the setup rather than with an inspection method and this is one of them. Concaves up to about a 3" radius can be cut. (See Figures 4 and 5.) Remember that when the cutting tool is extended a long way from the support of the yoke, it can be more difficult to control. Lighter cuts must be taken to achieve a good finish and accurate size but the tool should be controlled in a positive manner. Don't let the tool set on the part without cutting. Use the various pivot holes to try to keep the point of the tool as close to the yoke as possible to maximize the rigidity of your setup.

When cutting a concave radius you will use the holes nearer the center or short end of the yoke. For smaller radii, the cutting tool points into the "U" of the yoke. For larger radii the tool can be reversed and pointed toward the outside of the "U". In deciding which center hole to use it will help you to know that the center pivot hole is centered on the inside surface of the "U" and that the pivot holes are located on .250" centers.

Here is a simple formula that can also be useful when working with concave shapes:

$$r = \frac{c^2 + 4h^2}{8h}$$

where r = radius, c = diameter of pocket and h = height (depth of pocket)

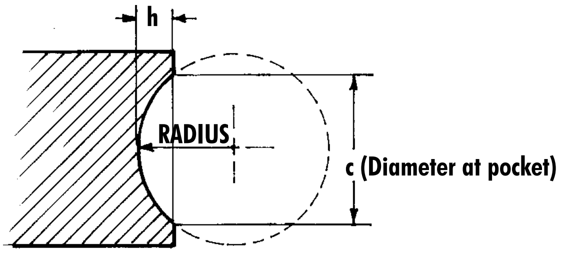


FIGURE 6—Measure the diameter of your pocket to obtain dimension "c". Then measure the depth of your cut with a depth micrometer or with the depth rod of your caliper against a straight edge to obtain dimension "h". You now have the dimensions you need to accurately calculate the radius you have cut. The radius can be calculated to the same level of accuracy as your measuring technique.

Making a complete ball

Using the radius cutter you can cut past the vertical point to make more than a half of a circle. However, because the cutter body will eventually hit the chuck, steady rest or some other part of your setup, you cannot cut a complete ball. (This is a problem for conventional horizontal-swing radius cutters too.) You can still make a complete ball using this tool, but you will have to do it in two steps.

First, turn half or a little more than half of the ball to the final radius and cut it off leaving enough material to form the opposite side. Make a mandrel with a diameter about 2/3 of the final diameter of the ball. Cut an angle into the face that will allow the completed half to be centered on the mandrel and epoxied to it. After the epoxy has hardened the ball can be completed with light cuts. Once finished, the ball is broken off the mandrel. By measuring the part with the anvil of the micrometer on the previously machined surface and the spindle of the micrometer on the surface you just machined the completed dimension should be equal to the diameter. A ball should always measure the same in any direction.

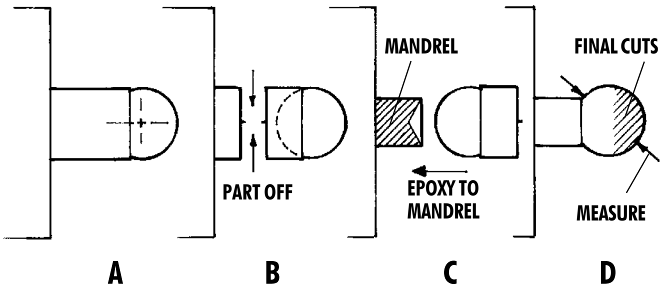
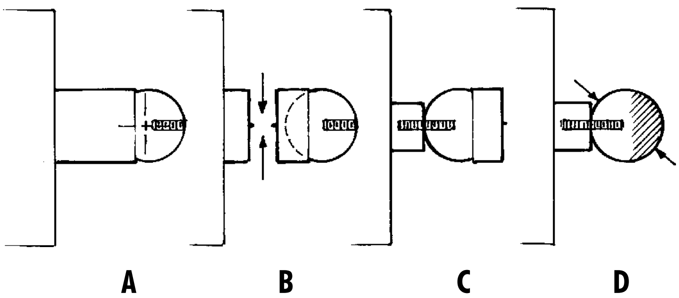


FIGURE 7—Cutting a complete ball using epoxy to attach the ball to a mandrel to complete the second half of the ball. A) Turn a little more than half the ball. B) Part off the piece leaving enough to complete the ball. C) Epoxy the piece into a mandrel with a tapered depression. D) Complete the ball using light cuts. Measure across the first and second turned portions of the ball to confirm the diameter.

Another method would be to center drill, drill and tap a hole in the end of the half-completed ball. Using a cutoff tool, part off the piece from the stock leaving sufficient material to complete the ball. Make a mandrel with a threaded stud centered on the end and screw your part onto it. Place the mandrel in the chuck and use the radius cutting setup you used to make the first half to complete the rest of the ball.

FIGURE 8—Using a threaded stud in the mandrel to hold

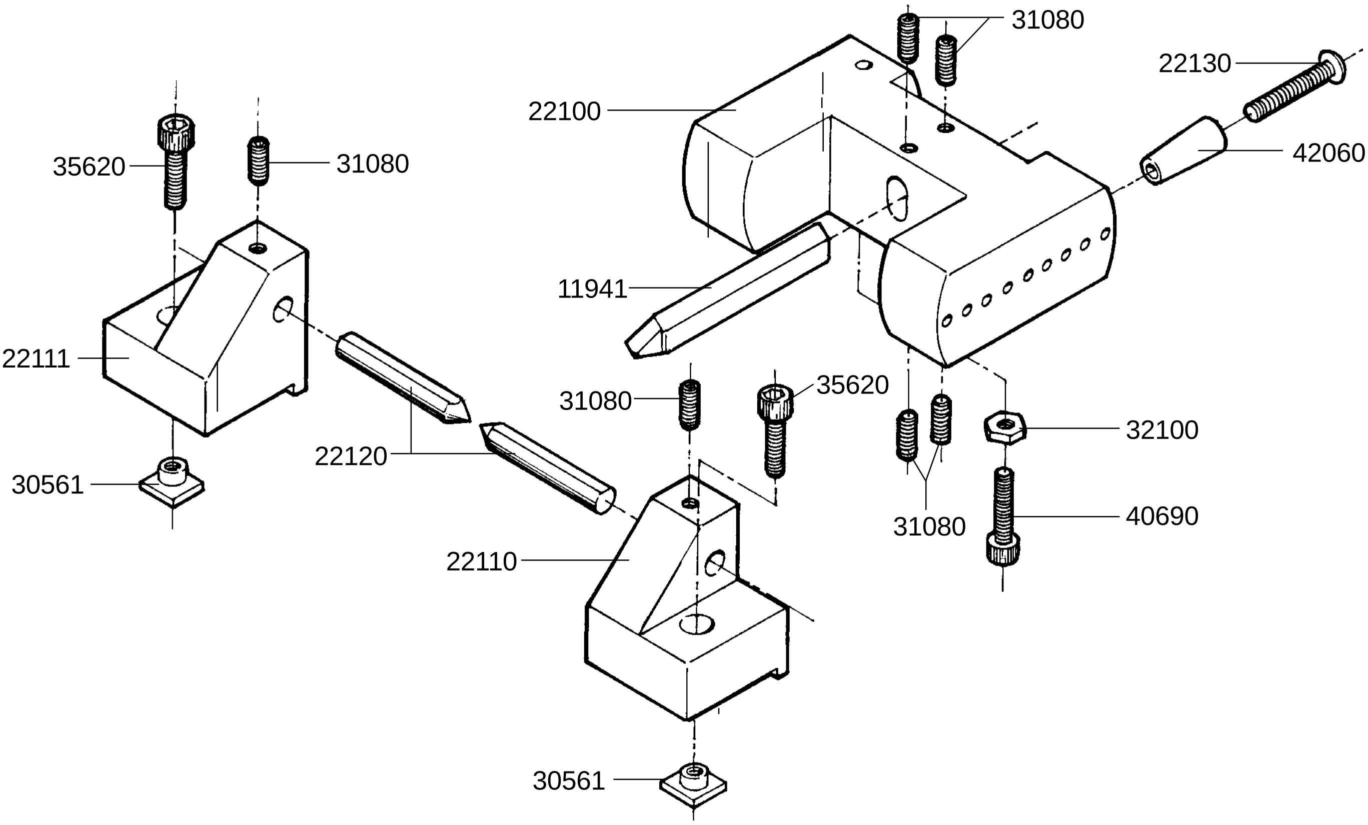


the ball for the second operation. A) Center drill the end and tap hole, then turn first half of ball. B) Part off. C) Attach part to threaded stud in mandrel. D) Turn final half of ball using light cuts and measure to confirm diameter.

The radius cutting attachment further extends the capabilities of your Sherline machine shop. With it you can apply a professional touch to your parts that would be difficult or impossible any other way. Though I have shown just a few examples here, I think you will find that, with a little imagination, there are many more ways it can be used.

—Joe Martin, President and owner
Sherline Products Inc.

EXPLODED VIEW



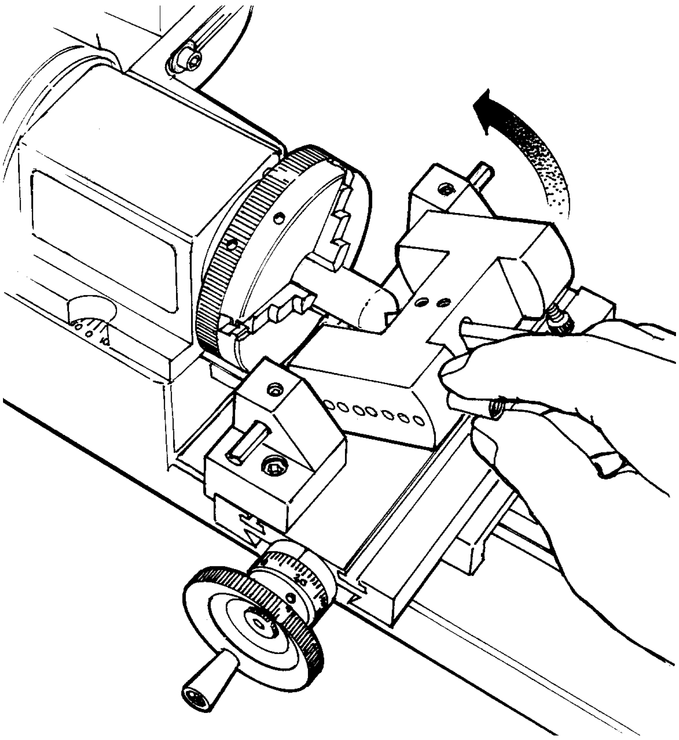
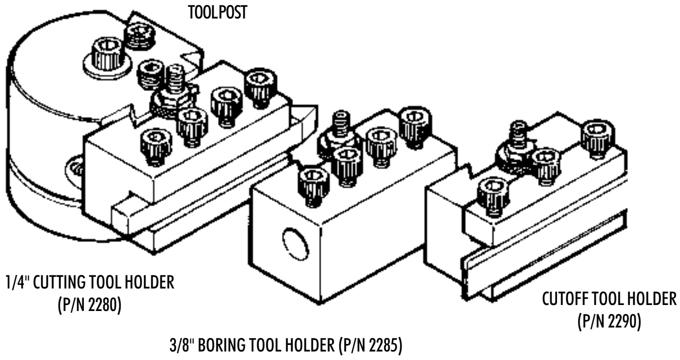


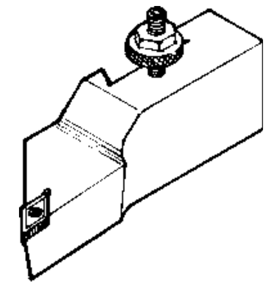
FIGURE 9—The radius cutter shown in use cutting a ball end on stock supported in a 3-jaw chuck.

PART NUMBERS AND DESCRIPTION

PART NO	NO REQ	DESCRIPTION
11941	1	1/4" Cutting tool
22100	1	Radius cutter body
22110	1	Radius cutter support (Left)
22111	1	Radius cuter support (Right)
22120	2	Radius cutter pivot pin
22130	1	10-32 x 1" button head socket screw
30561	2	10-32 T-nut
31080	6	10-32 x 3/8" cone point set screw
32100	1	10-32 hex nut
35620	2	10-32 x 7/16" SHCS
40690	1	10-32 x 3/4" SHCS
42060	1	Plastic handwheel handle

UNITurn & UNIMill
The tool tool





Optional P/N 2295 carbide insert holder.

QUICK-CHANGE TOOLPOST AND TOOL HOLDERS

P/N 2250 (And optional P/N 2295)

Purpose of the the quick-change toolpost

In a modern production machine shop, time is money, so being able to change tools quickly on a lathe becomes an economic necessity. Years ago a dovetailed holder and cam locking arragement was developed to make it possible to quickly change tools while leaving the body of the toolholder mounted to the table. For most users of tabletop size machine tools, however, the economic factor is not the prime reason for going to a quick-change tool system. Being able to change tools quickly simply means more time is spent doing the productive part of the job, and less time is wasted changing tools. The Sherline P/N 2250 quick-change set includes the toolpost and three holders: the P/N 2280 1/4" tool bit holder, the P/N 2285 3/8" boring tool holder and the P/N 2285 cutoff tool holder. An optional holder for inserted tip carbide tools is available as P/N 2295.

Construction of the quick-change toolpost and holders

The toolpost body and holders are machined from steel, case hardened and coated with a black oxide finish. This provides a very durable product that should provide generations of service if cared for. Two dovetails are provided on the toolpost so holders can be mounted in a choice of postions. The individual holders are locked in place with a cam against a dovetailed slide that is tightened using the same hex key used for many other Sherline operations.

Mounting the toolpost and changing holders

The toolpost is mounted to the lathe table using the same T-nut arrangement as the standard Sherline toolpost. A secondary clamp is also provided and may be used to insure the toolpost doesn't move. The horizontal flange of the angle clamp goes in the groove around the base of the toolpost. The tool holder is then slipped over the appropriate dovetail on the toolpost. The height of the tool tip is adjusted using the knurled handwheel. The cutting tip of the tool should be set to the centerline of the lathe by bringing the tool tip up to a Morse dead center in either the headstock or tailstock spindle. Once the height is set, lock the knurled wheel in

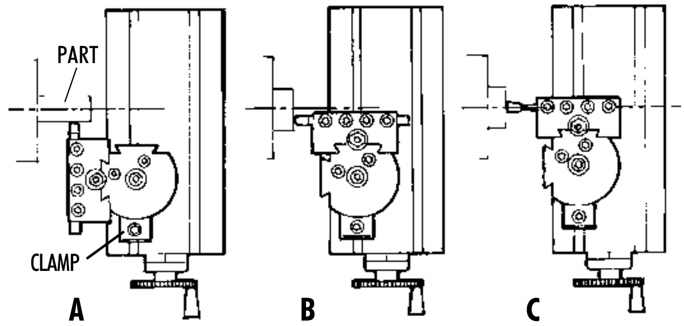


FIGURE 1—The 1/4" cutting tool can be mounted in either position. Figure A shows the positon for cutting and Figure B shows the position for facing. Figure C shows the boring tool positioned to bore the end of a piece of stock.

position with the hex nut. The holder can now be removed and replaced and the tool will remain at the proper height.

The boring tool holder

The boring tool holder will accept any boring tool with a 3/8" shank. This includes the 3/8" boring tools (P/N 3061and 3063) used with the P/N 3054 Sherline boring head. A variety of boring tool sets are available that would also fit the holder. These can be obtained from independent tool supply companies. It is a good idea to grind a small flat for the clamping screw to locate on. It isn't necessary to clamp all four screws to hold the tool. One or two will be sufficient. The additional screws are there to clamp a tool when the holder is used on the other dovetail.

Using a parting tool

After completing a part in the lathe it is frequently necessary to separate the part from the excess material used for chucking. This operation is best accomplished with the use of a cut-off tool or “parting tool” as it is sometimes called. The Sherline cut-off tool holder utilizes a very slender high speed tool steel cutting blade.The thinness of the Sherline blade (.040") enables it to feed into the part quite easily and at the same time minimizes the amount of waste material. If

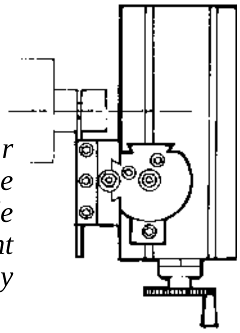


FIGURE 2—The cutoff tool or "parting" tool is held as shown. The height of the cutting tip of the blade should be adjusted using the height adjustment wheel until it is exactly on center with the spindle.

you use a non-Sherline blade, it is recommended that the blade be no wider than 1/16" when used in this holder. The turning speed for parting should be approximately one-half the normal turning speed for any given material. One word of caution; never use a parting tool on a part mounted between centers. The part may bind on the cutter and result in a scrapped part or a broken cutting tool.

Always try to lay work out so the cut-off tool is used as close to the spindle as possible. Set blade height using the adjustment wheel on the toolholder. It should be set so the tip is aligned with the centerline of the part being cut.

NOTE: ALWAYS USE CUTTING OIL WHEN USING THE CUT-OFF TOOL. The cut will be made much smoother, easier and cooler.

Speed should be slower than normal turning speed and feed rate should be a little heavy so the chip will not break up in the slot. If speed and feed are correct, there will not be any chatter, and the chip will come out as if it were being unrolled from a spool. Coolant (cutting oil) plays a major roll in this occurring properly.

If the tool chatters, first check to see if the work is being held properly. Then decrease speed (RPM) or increase feed rate or both. Once the blade has chattered, it leaves a serrated finish which causes more chatter. Sometimes a serrated finish can be eliminated by turning the spindle off, adding a liberal amount of cutting oil, bringing the blade up so there is a slight pressure on it without the spindle turning, and then turning by hand or as slowly as possible with the speed control to remove the chatter marks.

Sharpening Instructions

To sharpen the blade, set the tool support on the grinder in such a way that it will produce a 7° to 10° angle on the blade (top to bottom). (See Figure 3.)

FIGURE 3—Side view of blade.

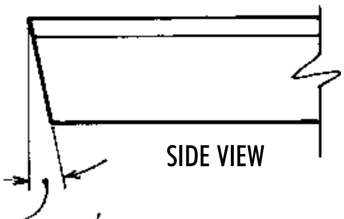
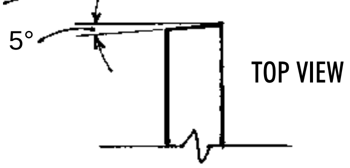


FIGURE 4—Top view of blade (enlarged) when ground for "parting off".



If you are sharpening the blade to "part off", the blade should have an additional angle of approximately 5° when viewed from the top with the point on the right. (See Figure 4.) Normally the angle would be as high as 15° but the .040" thickness of the blade would not be rigid enough and the blade could bend. If you want to cut grooves, don't put any angle on the blade when seen from the top.

If the cutting edges on the sides get dull, grind off the end of the blade until you get into new material where the edges are sharp to the cutting end. New blades are available as P/N 3086 from Sherline Products for $15.00.

Care of your toolpost and holders

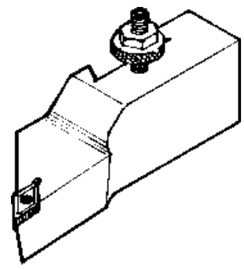
The case hardened and black oxide-finished steel components are very tough, but the corners and dovetail edges can be chipped if dropped or banged together. They should be protected from each other in your tool drawer. Because they are steel, they should also be protected with a light coating of rust preventative before putting them away.

Cutting tools available

Cutting tools are available from Sherline for the holders as follows:

- 1/4" cutting tool blank (P/N 3005) or a package of five blank 1/4" tools (P/N 3005B)
- 1/4" brazed tip carbide cutting tool set of 3 tools: left, right and 60° threading (P/N 3006)
- 1/4" pre-sharpened HSS cutting tool set of 3 tools: left, right and boring tool (P/N 3007)
- Cutoff tool blade (P/N 3086)
- 3/8" diameter boring tool holder that takes 55° inserted carbide tips (P/N 7635) or 3/8" diameter boring tool holder that takes 80° carbide inserts (P/N 7638)

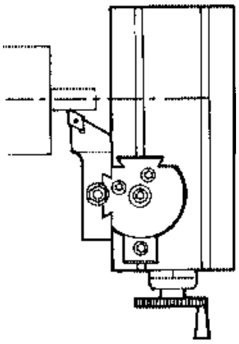
—Joe Martin, President and Owner
SHERLINE Products



Using the optional P/N 2295 inserted tip carbide cutting tool holder

For those who wish to take advantage of thesuperior cutting ability of carbide cutting tools, Sherline offers a tool holder for 55° inserted carbide tips. The P/N 2295 holder comes with one carbide insert which has 2 cutting edges. It is held in place with a small screw. A special Torx wrench to tighten the screw is also included. Like the P/N 2250 toolpost and three dovetailed holders, the inserted tip holder is manufactured from steel, case hardened and given a black oxide finish.

FIGURE 5—The inserted tip holder can be used for cutting as shown or for facing as shown in figure 1B on the first page.

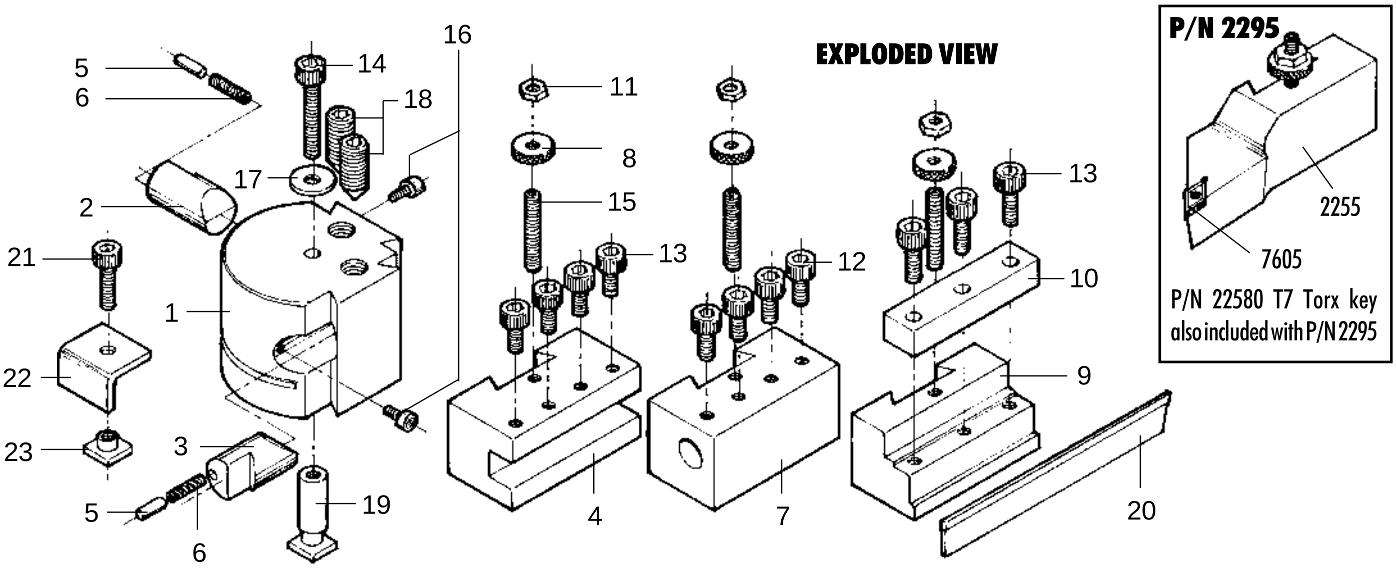


Replacement carbide inserts

Replacement inserts for this holder are available from Sherline as well as from several other tool manufacturers. Sherline's replacement number for a single carbide tip is P/N 7605. They are also available in a box of ten as P/N 7605B.

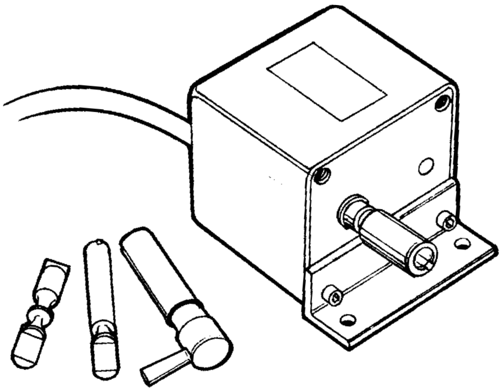
Riser block available

When using the quick-change tool post with the riser blocks in place on the lathe, a 1.25" riser block is now available. Ask for P/N 2251.



QUICK-CHANGE TOOLPOST PARTS LIST

REF NO.	NO. REQ.	PART NO.	DESCRIPTION	REF NO.	NO. REQ.	PART NO.	DESCRIPTION
1	1	22510	Toolpost body	13	7	40670	10-32 x 1/2" SHCS
2	1	22520	Left hand cam	14	1	40340	10-32 x 1" SHCS
3	1	22530	Right hand cam	15	3	22560	10-32 x 1" cone point set screw
4	1	22600	1/4" tool bit holder	16	2	22570	4-40 x 1/4" SHCS
5	2	22620	Cam spring pin	17	1	40660	3/16" I.D. washer
6	2	22630	.120" x 7/16" springs	18	2	40540	5/16" x 3/4" cone point set screw
7	1	22650	3/8" boring tool holder	19	1	40250	Extended toolpost T-nut
8	3	22680	Knurled height adjustment nut	20	1	30860	Cutoff tool blade (sold separately)
9	1	22700	Cutoff tool holder body	21	1	40330	10-32 x 5/8" SHCS
10	1	22710	Cutoff tool holder clamp	22	1	35580	Hold-down clamp
11	3	32100	10-32 hex nut	23	1	30560	10-32 T-nut
12	4	40510	10-32 x 3/8" SHCS				



UNITurn & UNIMill
The cool tool

POWER FEED

P/N 3001 (120V.)— P/N 3011 (240V.)

Reducing the diameter of a long shaft or a long part can be a tedious task requiring a lot of turning on the feedscrew. Obtaining a good finish on such a part requires slow, steady movement on the cutting tool, something hard to achieve when feeding the tool by hand. The UNITurn Power Feed was developed to eliminate this problem. A clutch mechanism permits quick disengagement of the motor so that you can hand feed the cutter whenever you desire. The power feed is from right to left at a constant (non-adjustable) speed of approximately 1.00 inches per minute. This speed was carefully selected and is appropriate for virtually all jobs you might want to do, making an expensive variable speed control unnecessary.

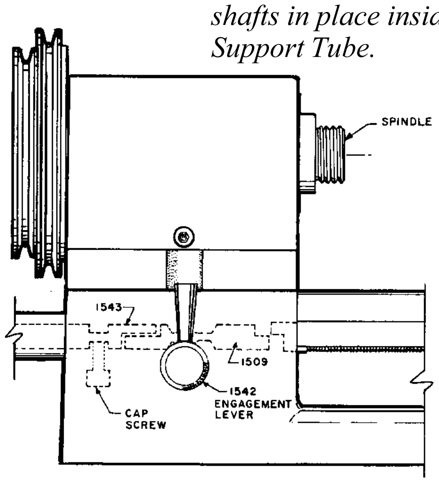
It is important to realize that the feed is an independent drive with a constant speed; whereas the spindle speed can vary. If spindle R.P.M. lowers, the cut becomes heavier, which in turn lowers spindle R.P.M. even more. As you can see, the end result could bind up the machine and bring it to a stop. Always bear this in mind when using this unit. If spindle speed starts dropping from too heavy a cut, disengage the feed drive first, then either take a lighter cut (approximately .015" in aluminum) or speed up the motor.

MOUNTING INSTRUCTIONS

1. Remove the headstock, the flat head socket screw under the headstock, and the socket head cap screw under the base.
2. Grease the shaft with flats on both ends (P/N 1509) and slide shaft into the protruding lead screw support tube situated directly below the main spindle pulley. Ensure end with small flat enters first. Now slide shaft with a single flat (P/N 1543) into the lead screw support tube. To guarantee that the shaft is "home", turn shaft one or two revolutions while applying gentle inward pressure to end of shaft. (See Figure 1.)

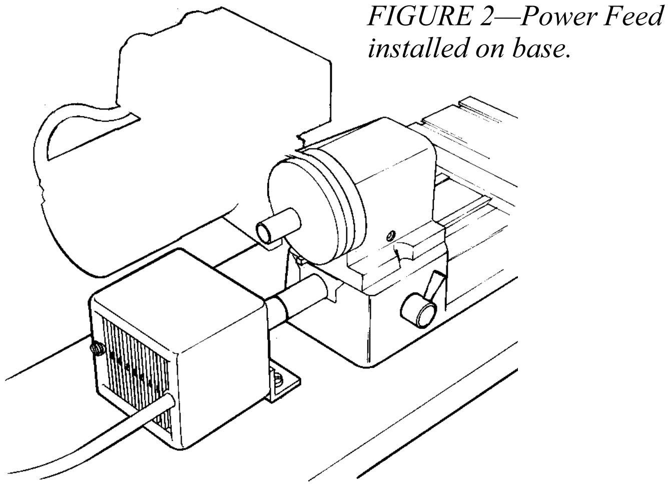
NOTE: If insertion or movement of the Engagement Lever is difficult, try loosening the two screws on the bottom of the machine that hold the bed to the base. Move the bed slightly until a good fit occurs.

FIGURE 1—Lead Screw engagement shafts in place inside Lead Screw Support Tube.



3. Replace screws removed in Step 1, making sure that point of screw goes into machined groove, and check that shaft from Step 2 is free to rotate.
4. Pull out black plug button (below nameplate) on side of lathe and slide shaft of Engagement Lever (P/N 1542) into hole, handle facing upward. It may be necessary to rotate shaft about 30° backwards and forwards to get it to engage properly.
5. Engage shaft of Power Feed unit and mount with bolts or sheet metal screws to same base as Lathe so shafts line up.

FIGURE 2—Power Feed installed on base.

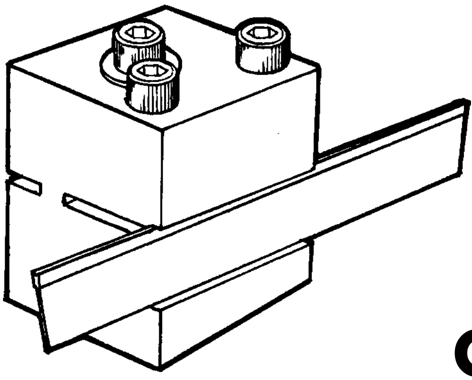


The cool tool®
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18
www.thecooltool.com

POWER FEED PARTS LIST

NO. REQ.	PART NO.	DESCRIPTION
1	1509	Sliding Shaft
1	1541	"O" Ring
1	1542	Engagement Lever
1	1543	Fixed Shaft
2	4051	Skt Hd Cap Screws, 10-32 x 3/8"
1	4052	Cup Pt Set Screw, 10-32 x 3/16"
1	4509	Sheet Metal Screw, #4 x 1/4"
1	4510	Power Feed Bracket
1	4511	Power Feed Cord w/Switch (U.S.A.)*
1	4512	Power Feed Motor Case
1	4513	Power Feed Motor (110V.)
1	4514	Power Feed Coupler
1	4063	Power Cord (U.K.)*
1	4064	Power Cord (Europe)*
1	4521	Rotary On/Off Switch (240V.)*
1	4525	Power Feed Motor (240V.)

*NOTE: U.S. models (P/N 3001) come with a rocker type on/off switch on the power cord, while U.K. and European models (P/N 3011) come with a toggle type on/off switch mounted on the side of the motor case.



UNITurn & UNIMill
The cool tool

WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

CUT-OFF TOOL AND HOLDER

P/N 3002

After completing a part in the lathe it is frequently necessary to separate the part from the excess material used for chucking. This operation is best accomplished with the use of a cut-off tool or "parting tool" as it is sometimes called. The UNITurn Cut-off Tool and Holder consists of a very slender high speed tool steel cutting blade mounted in a special tool holder. The thinness of the blade (.040") enables it to feed into the part quite easily and at the same time minimizes the amount of waste material. The turning speed for parting should be approximately one half the normal turning speed for any given material. One word of caution; never use a parting tool on a part mounted between centers. The part may bind on the cutter and result in a scrapped part or a broken cutting tool.

INSTRUCTIONS FOR USE

Always try to lay work out so the cut-off tool is used as close to the spindle as possible. Set blade height by sliding the blade in its slot in the tool holder. It should be set so the tip is aligned with the centerline of the part being cut. An unusual diameter may require a shim to be placed under the front or rear of the holder to accomplish this.

NOTE: ALWAYS USE CUTTING OIL WHEN USING THE CUT-OFF TOOL. The cut will be made much smoother, easier and cooler.

Speed should be slower than normal turning speed and feed rate should be a little heavy so the chip will not break up in the slot. If speed and feed are correct, there will not be any chatter, and the chip will come out as if it were being unrolled. Coolant (cutting oil) plays a major roll in this occurring properly.

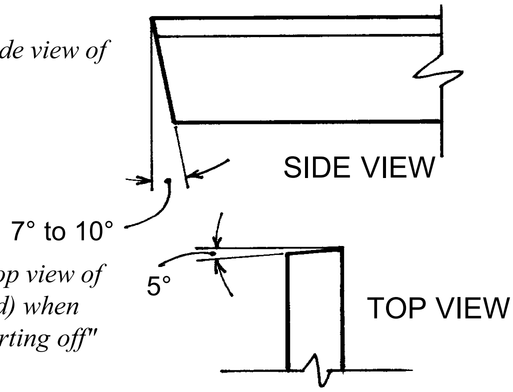
If the tool chatters, first check to see if the work is being held properly. Then decrease speed (RPM) or increase feed rate or both. Once the blade has chattered, it leaves a serrated finish which causes more chatter. Sometimes a serrated finish can be eliminated by turning the spindle off, adding a liberal amount of cutting oil, bringing the blade up so there is a slight pressure on it without the spindle turning, and then turning by hand or as slowly as possible with the speed control.

SHARPENING INSTRUCTIONS

To sharpen the blade, use the tool support on the grinder set in such a way that it will produce a 7° to 10° angle on the blade (top to bottom). (See Figure 1.)

FIGURE 1-- Side view of blade

FIGURE 2-- Top view of blade (enlarged) when ground for "parting off"



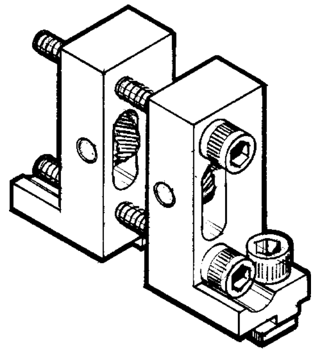
If you are sharpening the blade to "part off", the blade should have an additional angle of approximately 5° when viewed from the top with the point on the right (See Figure 2). Normally the angle would be as high as 15° but the .040" thickness of the blade would not be rigid enough and the blade could bend. If you want to cut grooves, don't put any angle on the blade when seen from the top.

If the cutting edges on the sides get dull, grind off the end of the blade until you get into new material where the edges are sharp to the cutting end. New blades can be purchased as Part Number 3086 and are available from The cool tool.

REPLACEMENT PARTS LIST

NO. REQ.	PART NO.	DESCRIPTION
1	3085	Cut-Off Tool Holder
1	3086	Cut-Off Tool Blade
1	4025	Tee Nut
1	4066	3/16" Washer
1	4071	10-32 x 1-1/4" Skt. Hd. Cap Screw
2	4074	10-32 x 7/8" Skt. Hd. Cap Screws

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18



WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

UNITurn & UNIMill
The cool tool

KNURLING TOOL AND HOLDER

P/N 3004

The knurling holder from manfred heindl is designed to be used only with the UNITurn lathe. The largest diameter that can be knurled is 1" (25.4 mm) and the smallest diameter is somewhat dependent on the pitch on the knurl. The higher the number of teeth per inch (TPI) the finer the knurl will be and the smaller the diameter that can be knurled.

The cool tool includes a set of basic knurls (25 TPI) that will produce a medium diamond knurled pattern. This set is a left and right pair with a 30 degree helix with each tool forming 1/2 of the diamond pattern.

A good knurl is produced by embossing; therefore, a correct starting diameter on the work to be knurled can best be determined by trial and error on a scrap piece of similar material. When knurls are forming they should be considered similar to one gear meshing with another. Think what would happen if you tried to mesh a 25 tooth gear with a gear that had 62 and 1/2 teeth. This is in effect what happens if the initial diameter is wrong causing the tools to take a second path every other revolution. This produces an undesirable finish.

The good part is that knurls have an amazing tolerance for wrong diameters when working with soft materials, and you will have better than an 70% chance of success on any given diameter.

Hard materials such as stainless and hardened tool steels will have short tool (knurls) life. Never attempt to knurl hardened material, such as piano wire.

The knurling holder is designed to mount directly to the crosslide's tee slot groove. The tee nuts that run in the groove should only be tightened enough to eliminate "play", but not so tight as to keep the holder from moving freely in the groove. This allows the holder to self center on the part to be knurled. (We recommend using aluminum for your first practice knurl, approximately 1/2").

The part to be knurled or experimental part should be running true with a chamfered corner at the end of the knurled section. Adjust the top and bottom clamping bolts so the knurls are lightly touching the part without the spindle turning. The knurls should be located at the beginning of the section to be knurled. Apply a liberal amount of cutting oil to the knurls and have the spindle run at a slow speed (approximately 500 RPM for 1/2" diameter of soft material).

Now start tightening the top and bottom clamping bolts evenly, one at a time until the knurls are engaged with the work in a positive manner. Back the knurls off the part with the feed handwheel. Stop the spindle and carefully examine the quality of the knurl. It should be full depth,clean, and sharp. The finished diameter should be approximately (see chart) over the starting diameter. If not make the necessary adjustments.

If the knurl isn't full depth take in on clamping bolts with the spindle running until it is full depth. (If the knurls are undercutting the finished diameter, the diameter should be either increased or decreased by approximately .005" (0.1 mm) until the knurls are working properly.) Increasing the diameter will add a tooth to the part. Decreasing the diameter will eliminate the knurls from having to remove so much material. If the knurl isn't clean and sharp use more cutting oil and turn the spindle slower and increase the feed.

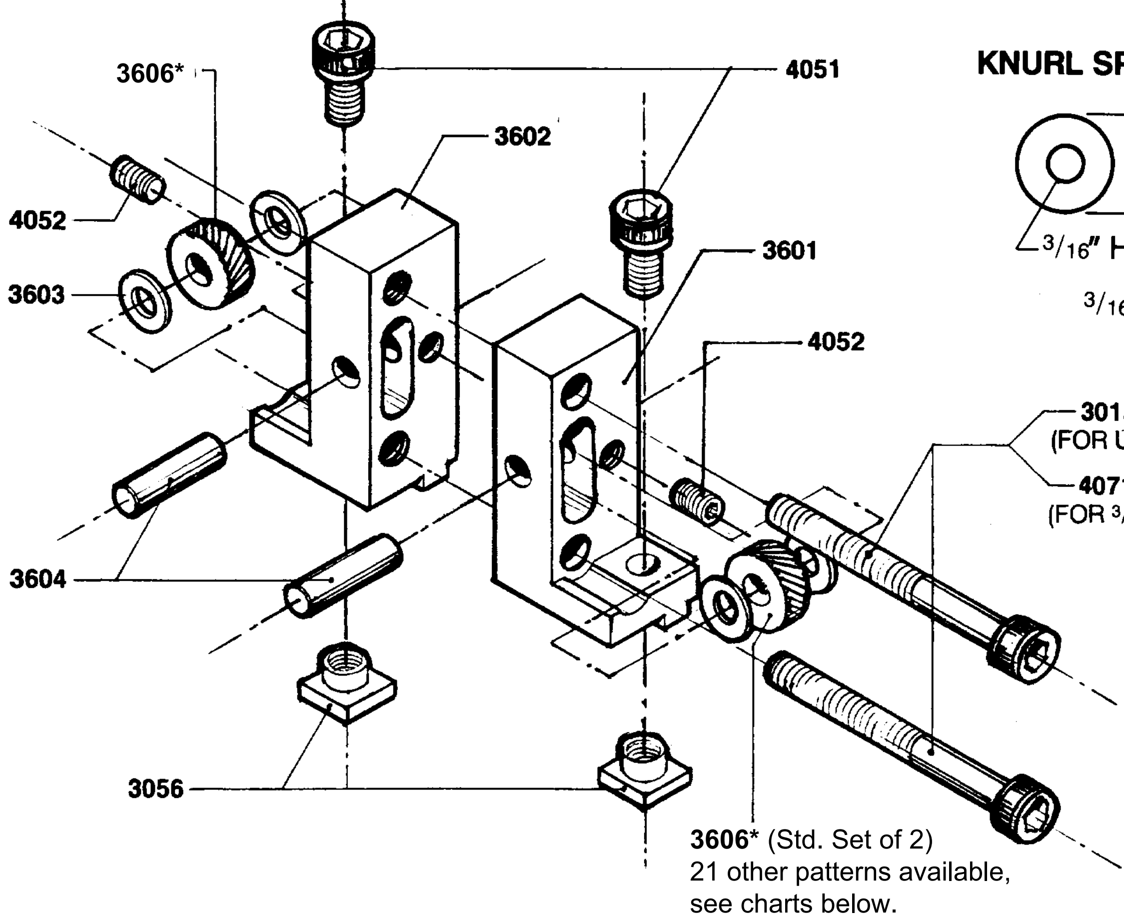
As you can see it isn't an exact science because of the many variables involved and that is why we recommend getting good results on a scrap part before attempting knurling a part you have a lot of work in.

Straight knurls have to be more carefully selected for the job if they are to be used for enlarging a shaft diameter for a press fit. In closer tolerance production work special knurls have to be made to accomplish this so the finer the knurl the better your chances of success are.

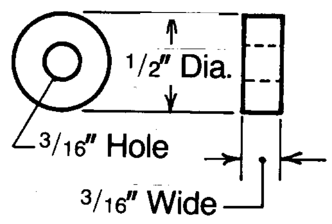
To complete the knurl the knurls are fed on to the part for the proper distance using plenty of cutting oil. Back the knurls off the part still using plenty of oil and you should have a knurl you can be proud of.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

EXPLODED VIEW



KNURL SPECIFICATIONS



OPTIONAL CIRCULAR PITCH KNURLS

TPI=Threads Per Inch/T=Teeth on the knurl

STRAIGHT TOOTH KNURLS

PART NUMBER	TOOTH ANGLE	TPI / T	QTY
3612	90°	16 TPI / 25T	PR.
3613	90°	20 TPI / 31T	PR.
3614	90°	25 TPI / 38T	PR.
3615	90°	30 TPI / 47T	PR.
3616	90°	32 TPI / 49T	PR.
3617	90°	35 TPI / 55T	PR.
3618	90°	40 TPI / 63T	PR.
3619	90°	41 TPI / 65T	PR.
3620	90°	47 TPI / 73T	PR.
3621	70°	35 TPI / 55T	PR.
3622	70°	50 TPI / 79T	PR.
3623	70°	53 TPI / 83T	PR.
3624	70°	60 TPI / 94T	PR.
3625	70°	60 TPI / 109T	PR.
3626	70°	80 TPI / 125T	PR.

30° SPIRAL KNURLS

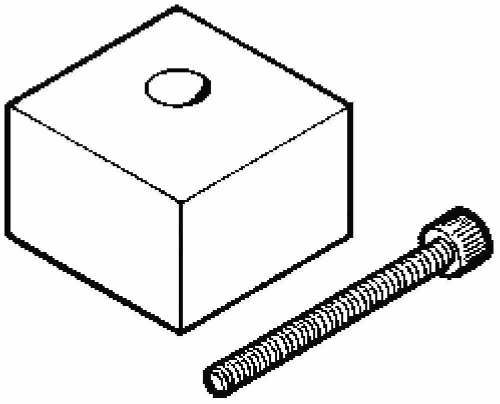
PART NUMBER	TOOTH ANGLE	TPI / T	QTY
3605	90°	20 TPI / 27T	PR.**
3606*	90°	25 TPI / 34T	PR.**
3607	90°	30 TPI / 40T	PR.**
3608	90°	35 TPI / 47T	PR.**
3609	90°	40 TPI / 55T	PR.**
3610	70°	50 TPI / 68T	PR.**
3611	70°	80 TPI / 107T	PR.**

*INCLUDED AS STANDARD WITH TOOL.

**PAIR INCLUDES ONE LEFT HAND AND ONE RIGHT HAND KNURL USED TOGETHER TO MAKE A DIAMOND PATTERN.

APPROXIMATE INCREASE IN KNURLED DIAMETERS

TPI	TOOTH ANGLE	STRAIGHT	DIAGONAL	DIAMOND MALE	DIAMOND FEMALE
12	90°	.034	.034	.038	—
16	90°	.025	.025	.029	—
20	90°	.020	.020	.023	.014
25	90°	.016	.016	.018	.011
30	90°	.013	.013	.015	.009
35	90°	.011	.011	.013	—
40	90°	.009	.009	.010	.006
35	70°	.014	—	—	—
40	70°	.012	.010	—	—
50	70°	.009	.009	.010	.006
60	70°	.007	.007	—	—
70	70°	.006	.006	—	—
80	70°	.005	.005	—	—



UNITurn & UNIMill
The cool tool

REAR MOUNTING BLOCK

(For P/N 3002 CUT_OFF TOL AND HOLDER)
P/N 3016

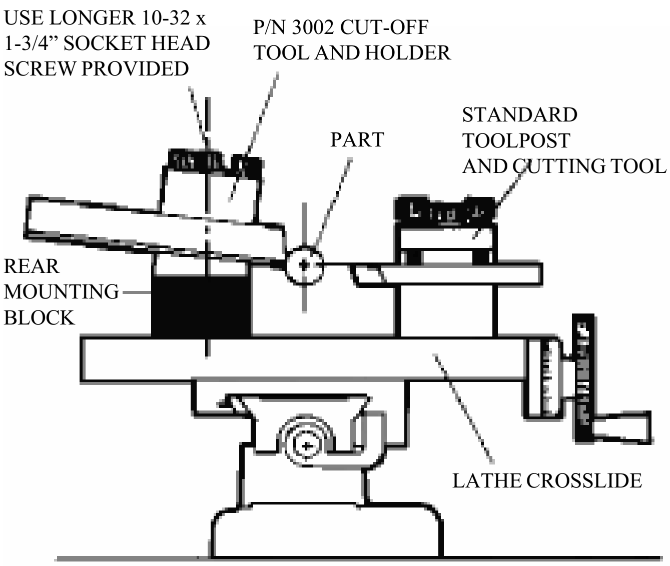


FIGURE 1—
(Looking from headstock toward tailstock) The rear mounting block is shown in position under the cut-off tool holder and mounted to the "back" side of crosslide table. The cut-off tool holder can now be left mounted to the table, ready for a parting operation at any time. There is no need to remove the standard tool post in order to part off the work.

NOTE: ALWAYS USE CUTTING OIL WHEN USING THE CUT-OFF TOOL.

See P/N 3002 Cut-Off Tool instructions for more details on use and sharpening of the tool.

Purpose of the Rear Mounting Block

The rear mounting block is a simple spacer block that allows you to mount the cut-off tool and holder to the table on the back side of the part. Because the part is rotating "up" on the back side, the tool must be flipped over in the holder. The mounting block raises the tool holder the amount needed to put the tip of the tool back at the right cutting height. This will save you time by being able to leave the cut-off tool holder mounted to the table while you use the regular tool post in its normal position on the front side of the part.

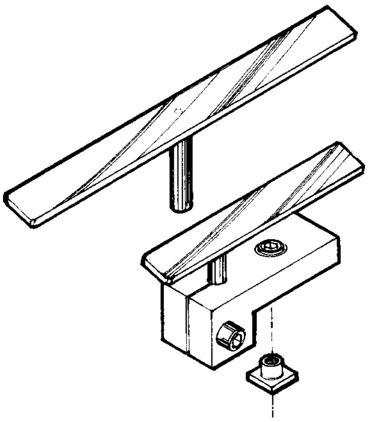
Instructions for Use

The mounting block is placed between the standard cut-off tool holder (P/N 3002) and the lathe crosslide. It is mounted on the back side of the part, or the side away from the crosslide handwheel. The longer 10-32 x 1-3/4" socket head screw provided with the rear mounting block is used to attach the unit to the crosslide table. (Use the T-Nut that came with the Cut-Off Tool Holder.) Note that the hole in the mounting block is not in the center. Rotate the block to find the position where the sides line up with the sides of the cut-off toll holder.

Loosen the two clamping screws which hold the cut-off tool blade in place. Turn the blade over so the cutting tip is facing down and mount it as shown in Figure 1. Adjust the tip of the tool to the desired height by sliding it back and forth in its slot. Lock it in position with the two clamping screws.

Refer to the instructions that come with the P/N 3002 cut-off tool for use. Even thought the tool is upside down, all the same rules about its use still apply. The only difference is that the crosslide table is now cranked *toward* the operator to cut off a part.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18



UNITurn & UNIMill
The cool tool

WOOD TOOL REST

P/N 3038
P/N 3047 (used with P/N 1291*)

The wood turning accessories are somewhat of an afterthought. A machine designed to cut metal can easily cut wood using the same methods as cutting metal. The machine can become a little cumbersome when used as a wood turning lathe. The tool support is mounted on the crosslide, and if you try to use it like a standard wood turning lathe, the crosslide and handwheel would be in your way. Work with your tool angled above center rather than below center as shown in most instructions on wood turning.

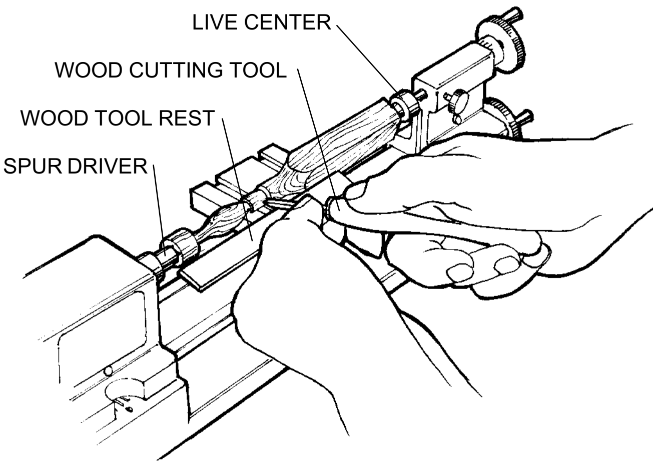


FIGURE 1—Holding the wood cutting tool.

This can eliminate the problems of long handled wood turning tools hitting the table.

Success will be determined by having good turning tools and good hardwoods, such as maple, to work with as the UNITurn has a superior spindle to that on wood turning lathes. There isn't any reason you can't cut wood with lathe tools, but they have to be very sharp. If you have a lot to cut you may have more success with more rake on your tools. Again, the harder the wood the better it will cut.

We also manufacture a Spur Driver (P/N 3035) to work between centers. A Live Center (P/N 1191) is also a must for this. When working with small diameters you have the advantage of holding work with the 3- or 4-Jaw chucks or even with the collets.

We added this accessory to our production line because we were constantly asked for something to make very small wooden parts. I visualized these attachments making small parts like doll house furniture when I designed it which is the reason for an offset support. This design allows you to work with the tailstock spindle close to the spindle and still get a support in between them.

We made a trial run of parts and the first call I received was the support arm was too short. The part he was making was a fishing rod handle which is why there are now two support arms included. The second one is for working on longer parts.

The next call was for an extended tool support shaft to be used with our Riser Blocks (P/N 1291 and 1292). A lamp was the project this time. An Extended Wood Tool Rest is now available as Part Number 3047.

For additional information on wood turning techniques, there are many fine books written for reference.

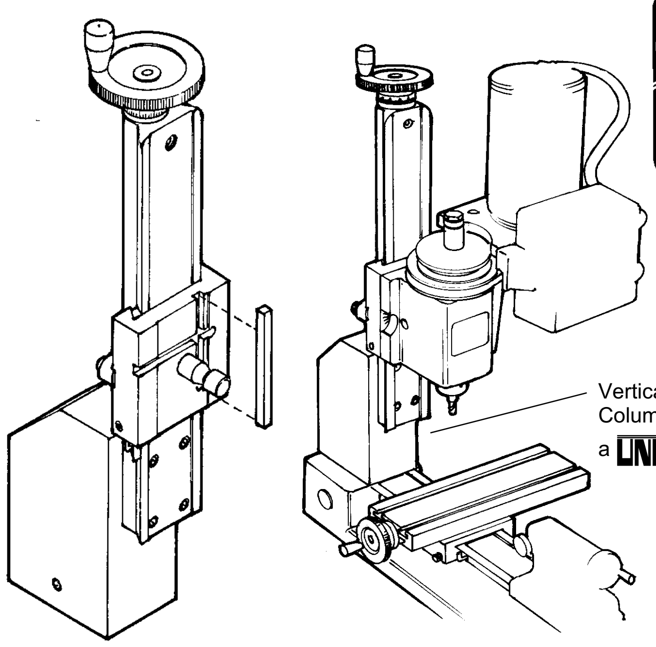
We hope you find these to be useful accessories for your UNITurn lathe.

REPLACEMENT PARTS LIST

NO. REQ.	PART NO.	DESCRIPTION
1	3044	Wood Tool Rest, 3" (use w/ P/N 1291)*
1	3045	Wood Tool Rest, 3"
1	3046	Wood Tool Rest, 5"
1	3048	Wood Tool Rest, 5" (use w/ P/N 1291)*
1	3039	Wood Tool Post Body
1	3056	Tee Nut
1	4069	10-32 x 3/4" Skt. Hd. Cap Screw
1	4077	10-32 x 5/16" Skt. Hd. Cap Screw

* P/N 1291 is a riser block kit. Parts No. 3044 and 3048 have extended shafts to accommodate the extra height.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone: +43-2236-892 666 fax: +43-2236-892666-18



Vertical Milling Column mounted on a UNITurn Lathe

WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

UNITurn & UNIMill
The cool tool

Refer to UNITurn INSTRUCTION GUIDE (P/N 5326) for milling Setup and Operations.

VERTICAL MILLING COLUMN

P/N 3050 (Inch), P/N 3053 (Metric)
P/N 3480 (Inch), P/N 3485 (Metric)
w/ Zero Adjustable Handwheel

With this attachment the UNITurn Lathe can be quickly and easily converted into a small milling machine. The attachment consists of a dovetailed vertical column with a solid aluminum base that attaches to the bed of the lathe in place of the headstock. The headstock then mounts to a dovetailed saddle on the vertical column. The saddle is raised and lowered to control the depth of cut by turning a handwheel. Calibrations on the handwheel enable depth control to .001 inch. Parts to be machined are mounted on the lathe's 2.75" x 6.00" crosslide.The headstock may be locked in position by means of a screw on the back of the saddle. (See P/N 4517 and 4033 on the exploded view.)

This is a good way to get into milling. All standard vertical milling operations can be performed with this attachment with size being the only limitation. Conversion from the lathe to the mill takes less than one minute. Just about all UNIMill DeLuxe -milling accessories may be used with this setup. (NOTE: Due to the size and weight of the P/N 3200 Indexing Attachment and the P/N 3700 Rotary Table, it is not recommended they be used with the Lathe and Vertical Milling Column combination. We recommend they be used with the UNIMill DeLuxe Vertical Milling Machine or XYZ Base.)

NOTE: All Vertical Milling Columns manufactured after 1995 include the modifications necessary to work with UNITurn's XY Base.

MOUNTING INSTRUCTIONS

Remove headstock from lathe by loosening set screw (P/N 4054) located below name plate. Lift the headstock vertically from the bed.

Mount the Vertical Milling Column on the lathe pivot pin (P/N 4024) on the lathe bed. Mount the headstock on the pivot pin located on the saddle of the Vertical Milling Column in the same manner as it is mounted on the lathe bed. Angles can be milled by rotating the headstock with the alignment key removed. Move headstock to desired angle and tighten the set screw (P/N 4054).

HELPFUL HINTS

1. This is a small, light duty mill, and should not be used to remove vast amounts of unnecessary stock that could be easily removed with a hacksaw. Get stock as close to size as possible before starting.
2. Loads involved for milling are higher than for lathe turning. The vibration level is also higher, therefore, more attention must be paid to gib adjustments. They should be kept snug.
3. End mills must run true and must be sharp. Holding end mills in a drill chuck is a poor method. Use milling collets or our end mill holders. For cutting aluminum, run motor top speed and take light cuts.
4. Fly cutting is an excellent way of cutting stock from flat surfaces.
5. Normal machine alignment is good for most work, but if the work is exceptionally large or has to be extremely accurate, shims may be required to improve machine alignment.
6. Learn to use a dial indicator.
7. A good vise is a must.
8. Often, more time will be spent making fixtures to hold work than doing the actual machining.
9. Always try to have one point to measure from. Don't machine this point off half way through the job and leave yourself without a way of measuring the next operation. **PLAN AHEAD**.
10. A good rule for all machining operations is,"if the tool chatters, reduce speed and increase feed."

It takes a long time to accumulate the knowledge, tools and fixtures to do the tremendous amount of different type of operations involved in milling. Don't get discouraged by starting with a job that is too complex or by using materials that are extremely difficult to machine.

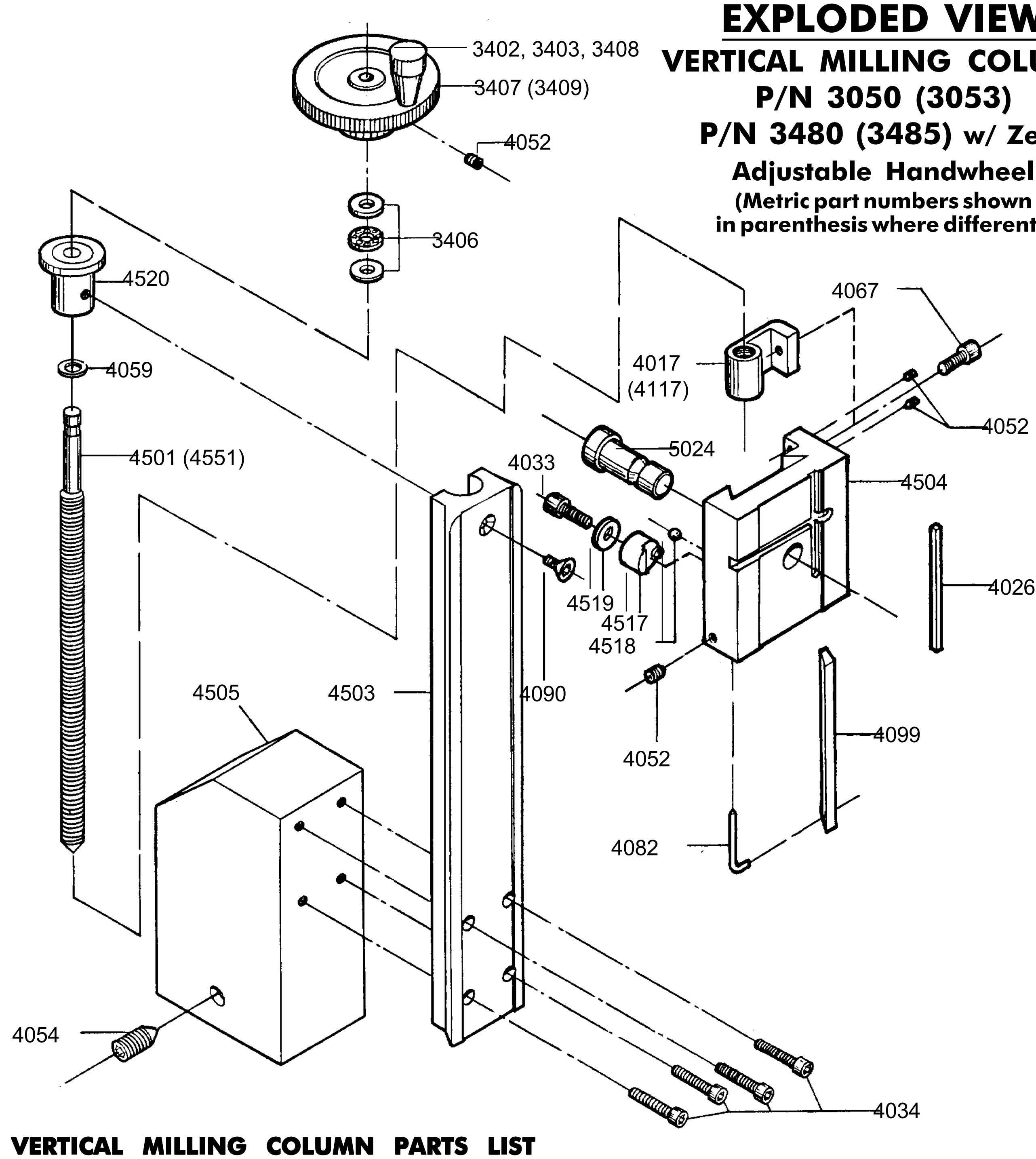
The cool tool®
www.thecooltool.com

Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

EXPLODED VIEW

VERTICAL MILLING COLUMN
P/N 3050 (3053)
P/N 3480 (3485) w/ Zero Adjustable Handwheel

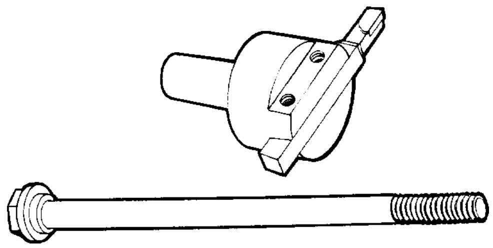
(Metric part numbers shown in parenthesis where different.)



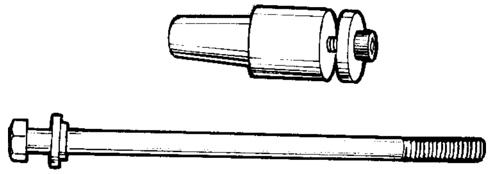
VERTICAL MILLING COLUMN PARTS LIST

NO. REQ.	PART NO.	DESCRIPTION
1	3402	"Z" Axis Handwheel Knob
1	3403	"Z" Axis Handwheel Shaft
1	3406	Thrust Bearing Set
1	3407 (3409)	"Z" Axis Handwheel Body (for P/N 3450)
1	3408	Handwheel Plug
1	3422	Lock Nut, Adjustable Handwheel (for P/N 3480/3485)
1	3425	Lock Screw, Adjustable Handwheel (for P/N 3480/3485)
1	3426 (3427)	"Z" Axis Zero Adj. Hndwhl. Collar (for P/N 3480/3485)
1	3441	"Z" Axis Zero Adjustable Handwheel Body (for P/N 3480/3485)
1	4017 (4117)	Saddle Nut
1	4026	Head Key
1	4033	Skt Hd Cap Screw 10-32 x 5/8"
4	4034	Skt Hd Cap Screws 10-32 x 1"
4	4052	Cup Pt Set Screws 10-32 x 3/16"
1	4054	Cone Pt Set Screw 5/16-24 x 3/4"
1	4059	Washer, 1/4" I.D.
1	4067	Skt Hd Cap Screw 10-32 x 1/2"
1	4082	Gib Lock
1	4090	Flat Head Screw 10-32 x 3/8"
1	4099	Saddle Gib
1	4501 (4551)	Column Lead Screw
1	4503	Column Bed
1	4504	Column Saddle
1	4505	Column Base
1	4517	Column Saddle Lock
1	4518	3/16" Ball Bearing
1	4519	#10 Washer, Type B
1	4520	Bored Column Thrust

UNITurn & UNIMill
The cool tool



FLYCUTTER P/N 3052



SLITTING SAW HOLDER P/N 3065

FLYCUTTER AND SLITTING SAW HOLDER

P/N 3052 AND P/N 3065

Both the Flycutter (P/N 3052) and the Slitting Saw Holder (P/N 3065) are held in the spindle with a drawbolt that pulls these holders up into the Morse #1 taper. This is a "sticking" taper and it has to be removed from the spindle by backing out the drawbolt a few turns (do not disengage) and giving the bolt a few light taps with a hammer.

A Flycutter is a great way to machine flat surfaces. It can be easily sharpened and is probably the most economical way to remove material on a mill. The cutter is basically a left handed lathe tool. We supply it with a carbide tip cutter, but there is no reason a piece of 1/4" square high speed steel wouldn't work.

As with all machining operations, it is imperative the work is securely held to the work table. A flycutter on the UNIMill DeLuxe can cut a path 2" (50 mm) wide by .010" (.25 mm) deep in aluminum without even trying. Flycutters exert lower machining stresses on the machine than you may expect. This is because the cutter "peels" the material off with very little crushing action. If possible the cutter should swing a diameter larger than the part width. The cutter will usually take a second cut with the back side of the cutter even when the spindle is perfectly square with the table.

Chips thrown off by the flycutter are **HOT**. Long sleeve shirts are advisable and eye protection is a must!

If you're machining aluminum, run the spindle at 1/2 speed, with steel use 1/4 speed, and use a feed rate that creates curly chips about .002" (.05 mm) thick. You really should have some understanding of cutting speeds if you use high speed steel cutters on steels. It is very easy to exceed the cutting speed of high speed steel with a large cutter diameter. An example would be a H.S. cutter swinging a 2" (50 mm) circle shouldn't exceed 200 RPM when cutting steel with a cutting speed of 100 surface Ft/Min.

$$\frac{\text{4 X CUTTING SPEED (Ft./Min.)}}{\text{CUTTER DIAMETER (In.)}}$$

$$\frac{4 \text{ X } 100}{2} = 200 \text{ RPM}$$

Note: The factor 4 used in this equation has been rounded off to allow mental calculations, the actual number should be 3.8

In metric a close approximation is:

$$\frac{\text{300 X CUTTING SPEED (M/Min.)}}{\text{CUTTER DIAMETER (MM)}}$$

Note: The factor 300 has been rounded off to simplify calculations, the actual number is 318.

The reason we go into cutting speed in these instructions is that they are such an important part of using slitting saws correctly. You must realize that when you exceed the cutting speed with high speed steels, the dulling process can be instantaneous. It isn't you get shorter tool life, you get "no life"! This can be expensive in time and money because slitting saws usually cost so much you don't have spares.

Another problem that happens with slitting saws is that one edge gets dull before the other. This causes the blade to deflect as it cuts. How much it deflects is somewhat a function of the blade's thickness.

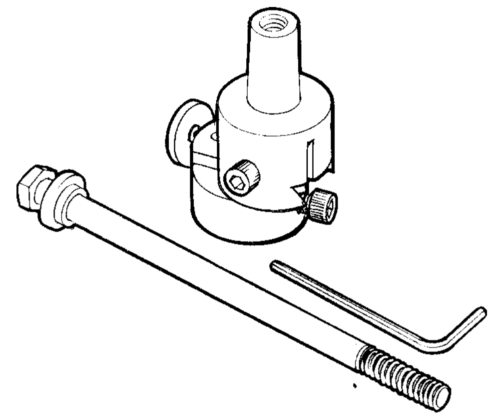
A common error that can be made is putting a slitting saw on the spindle upside down when they will only cut one way.

Coolant should be used. It doesn't have to be flooded, but it should be applied liberally to keep the fine teeth from loading up.

When cutting a slot that goes into a large hole, it is possible to have the part clamp the saw blade as it cuts through. It usually doesn't cause any serious problems, just stop the spindle and back it out.

There is always a question of the best approach to use when cutting a slot with a blade that is less than .060" (1.5 mm). You can cut in a series of passes, cut full depth in one pass, or cut straight in. The method to use is up to you, experiment with scrap until you're confident you will not screw up a lot of work.

Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18



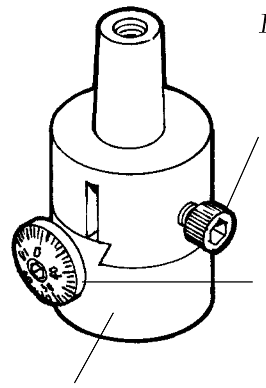
WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

UNITurn & UNIMill
The cool tool

BORING HEAD
P/N 3054

Boring holes on a mill is very similar to boring holes on a lathe except the cutting tool moves rather than the part. The main advantage of boring over drilling is that the hole will always come out in perfect alignment with the spindle whereas a drill may "wander". Larger holes must be bored rather than drilled on a small mill because large drills cannot be chucked in a UNITurn and it takes horsepower to drill holes over 3/8" (10mm).

Figure 1



Tools put in the UNITurn boring head should be as short as possible to keep set ups as rigid as possible. It is easier to bore a hole completely through a part than to cut into a flat bottom hole. The problem is tool chatter when you get to the bottom. A hole has to be started with a drill to the full depth of the finished hole. Many times the work will require a special boring tool that usually can be made from a standard boring tool.

Boring tools that are commercially available with a 3/8" shank have a shank that is too long for our boring head. To insure a rigid set up, cut part of the shank off. The shank of the tool should not extend much below the boring head. UNITurn boring tools are ready to use without this cutoff operation.

Boring holes over an inch deep can get difficult because it takes long tools which compromises rigidity. If you need a large flat bottom hole, consider doing it on a rotary table with an end mill.

If you have to have a flat bottom hole, a good tip is to turn the spindle off .002" (.05mm) from the bottom and rotate the spindle by hand while feeding the spindle down the remaining distance to eliminate chatter at the bottom. A wrench on the boring head drawbolt can make it easy to rotate while cutting.

If you have an existing hole's location out of tolerance, many times you can use a boring tool to correct the location. A boring tool follows the spindle, not the hole. A bushing can be made to press into the bored hole to correct the diameter that has been bored oversize. This method can also be used to correct shaft holes that have worn elliptically in space plates.

Remember the rule: "If a tool chatters, reduce speed (RPM), Increase Feed (Rate handwheel is turned) and take lighter cuts (Boring head adjustment)."

CAUTION! WORK MUST BE SECURELY CLAMPED TO THE TABLE.

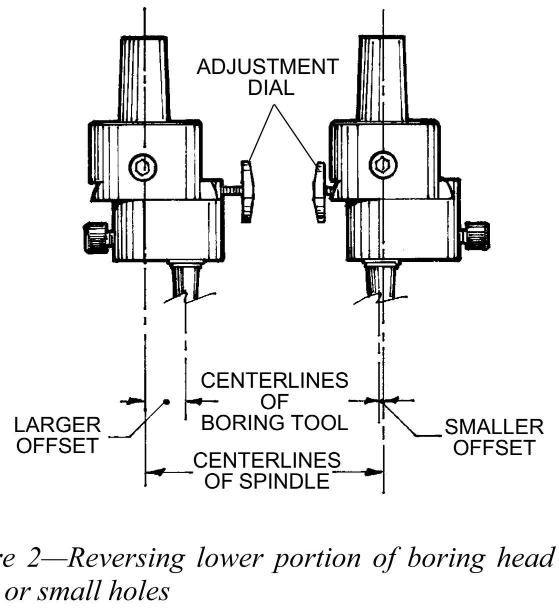


Figure 2—Reversing lower portion of boring head for large or small holes

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

Drill a hole as large as you can and still leave at least .060" (1.5mm) to finish bore. Decide the configuration of the boring head. The hole that accepts the boring tool can be indexed for large or small diameter holes. Use the combination that will keep maximum engagement of the boring head's dovetail..

Clamp the boring tool into the boring head so the cutting face of the tool is in alignment with the center of the spindle.

Adjust the tool in or out to take approximately .020" (.5mm) cut in aluminum.

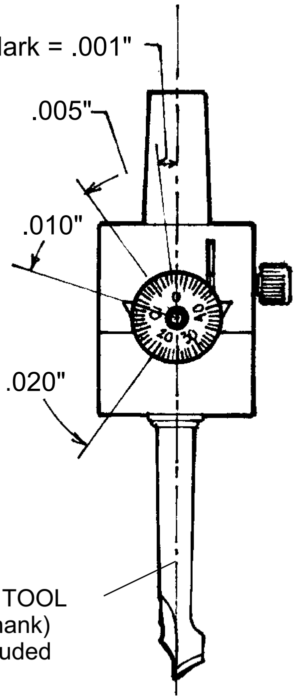


Figure 3—Fine adjustments of the Boring Head

One turn of the 4-40 screw dial will produce an increase of .050" in the bored diameter. Each increment on the dial is 7° which produces a rotation of the screw equivalent to .001" in the bored diameter.

Major increments are marked on the dial at 10 (.010"), 20 (.020"), 30 (.030") and 40 (.040").

Note: Do not attempt to bore any metal other than aluminum or brass until you have the skill to hold exact diameters on these easier to machine materials.

Be sure the boring head screws are tightened properly and turn on spindle at approximately 1/4 speed. Take a cut by feeding the tool into the part at a rate that keeps a continuous chip. Feeding too slow can cause chatter. A small amount of cutting oil will dramatically help the process.

Repeat this process until the hole is "roughed" out. Leave about .030" (.7mm) for finishing.

Before going to the finish diameter, determine that you can get a suitable finish with the tool you're using. Take a light cut and stop the spindle at the bottom and inspect the finish. Start the spindle and back the tool out. Usually the tool will take a light cut on the way out. Stop the spindle and inspect the finish. What you should learn from this exercise is where to stop your cut for the best possible finish; in or out?

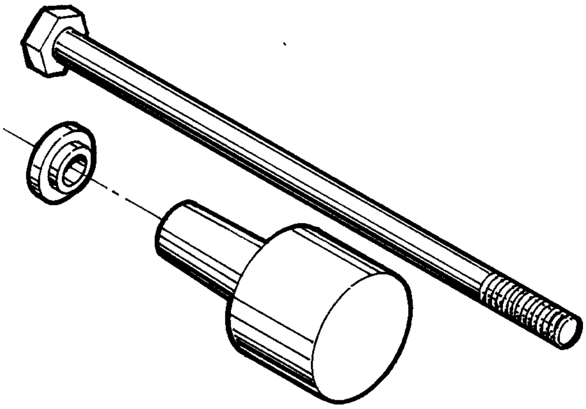
If your tool needs sharpening, sharpen it before you get near the finish diameter. You can't bore a hole any more accurately than you can measure it. Learn how to use small hole gauges and telescoping gauges. If you only have dial calipers, bore the hole and, if possible, turn the mating part on a lathe to fit the hole if a good fit is required (lathes are easier to hold tight tolerance diameters on).

From this point on it's best to "sneak up" on the finish diameter by taking half the cut required to get to the finish diameter. Cuts will keep getting smaller, and when you get to an error so small it would be hard to adjust the boring head .001" (.02 mm), try feeding the tool in at a higher RPM to bring the diameter to size.

Only the basics are written in these instruction and to make these basics work, it requires a liberal amount of common sense. If you have any doubts about a set up, it isn't good enough! The skill of machining is making accurate parts first try. If this is your first attempt to use a boring head, bore a hole .575" or 16mm through a piece of aluminum approximately 3/8" (10mm) thick and see if you can come within .0005" (.025mm) first try. This will be a good test of your machining skills!

REPLACEMENT PARTS LIST

NO. REQ.	PART NO.	DESCRIPTION
1	3088	1/4-20 x 5-1/8" Drawbar and Washer
1	3107	Gear Drive Pin
1	3154	Boring Head, Primary (top)
1	3155	Boring Head, Secondary (bottom)
1	3156	4-40 x 3/4" Skt. Hd. Cap Screw
1	3157	Adjustment Dial
1	4034	10-32 x 1" Skt. Hd. Cap Screw
1	4057	3/32" Hex Key
1	4069	10-32 x 3/4" Skt. Hd. Cap Screw



UNITurn & UNIMill
The cool tool

MORSE #1 BLANK

P/N 3055

The Morse #1 blank is made from free machining steel and is available so that you may make your own custom tool holders. Below are some drawings suggesting ways to make holders for end mills, fly cutters and slitting saws. The proper taper fit is already machined onto the tapered end. You need only drill, tap or slot the blank to fit your special tooling needs.

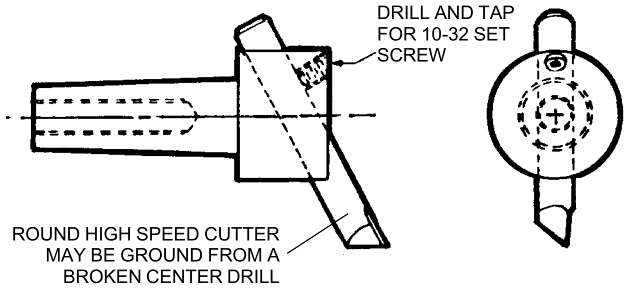


FIGURE 2--Making a Fly Cutte.

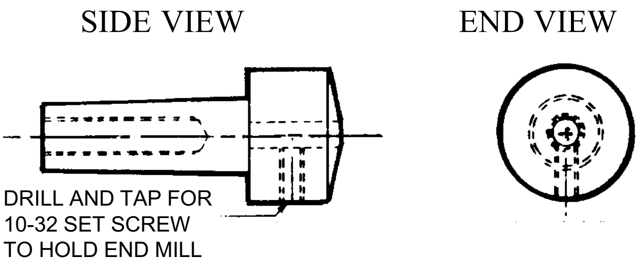
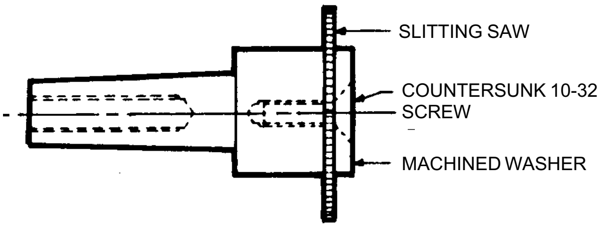


FIGURE 1--End Mill Holder



Making a Slitting Saw Holder

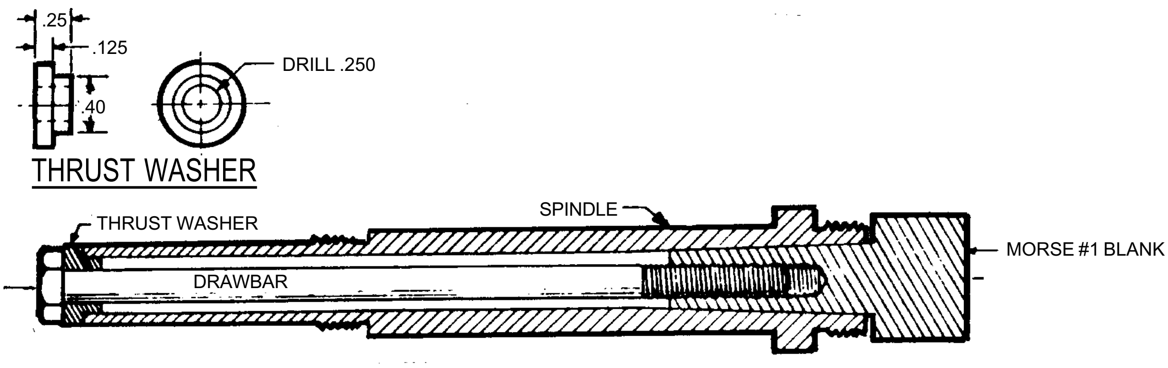
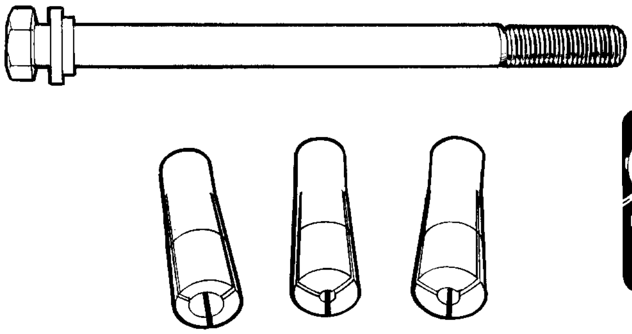


FIGURE 4--Cross-section of Lathe Spindle showing Morse Blank, Drawbar and Thrust Washer in position.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone: +43-2236-892 666 fax: +43-2236-892666-18



UNITurn & UNIMill
The cool tool

MILLING COLLETS

Collet Sets P/N 3060 (Inch), P/N 3090 (Metric)
End Mill Holders, P/N 3079 (3/8"), 3078 (10mm), 6079 (1/4"), 6080 (3/16")

The milling collets (P/N 3060) used with the UNIMill DeLuxe Vertical Mill or Vertical Milling Column are designed to be used with the Morse #1 taper common to all headstock spindles manufactured by UNIMill DeLuxe. The collets are held into the spindle with a drawbolt. The set includes 3 collets and a drawbolt with collar.

These collets have a shallow angle that give them high clamping pressure making them ideal for holding cutters. The shallow angle makes the collet "stick" after the collet drawbolt is loosened. Back the bolt off a few turns (do not disengage completely) and lightly tap the head of the bolt with a hammer or mallet until the collet can be easily removed.

Milling Collets are available in the following sizes:

P/N 3087	3/32" Mill Collet
P/N 3089	5/32" Mill Collet
P/N 3091	7/32" Mill Collet
P/N 3092	3.0mm Mill Collet*
P/N 3093	4.0mm Mill Collet*
P/N 3094	6.0mm Mill Collet*
P/N 3095	1/8" Mill Collet**
P/N 3096	3/16" Mill Collet**
P/N 3097	1/4" Mill Collet**

*Included with set P/N 3090
**Included with set P/N 3060

END MILL HOLDERS

Because the hole through the spindle is only a little over 3/8" (10mm), a collet that would accept a 3/8" shank end mill is impossible to make. End mills with 3/8" shank are very common and in many cases cost less than the miniature series. They are available in many sizes and shapes. To take advantage of this fact, UNIMill DeLuxe, offers the 3/8" End Mill Holder (P/N 3079). (See Figure 1.) Metric version of 3/8" End Mill Holder also available (10mm, P/N 3078).

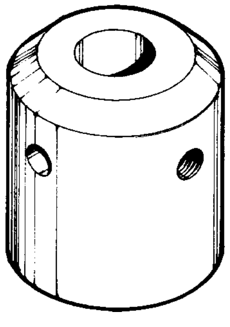


FIGURE 1— End Mill Holder.

The holder is manufactured on a modern CNC lathe allowing the internal 3/4-16 thread to be single pointed and without unchucking the part, accurately boring the 3/8" hole. This fact allows us to have the end mill run very accurately even though it is held on a threaded surface. Also available are 1/4" bore (P/N 6079) and 3/16" bore (P/N 6080) holders in this same style.

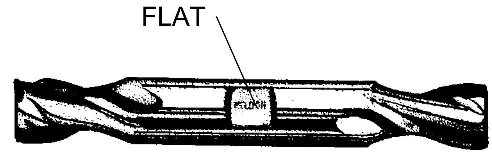
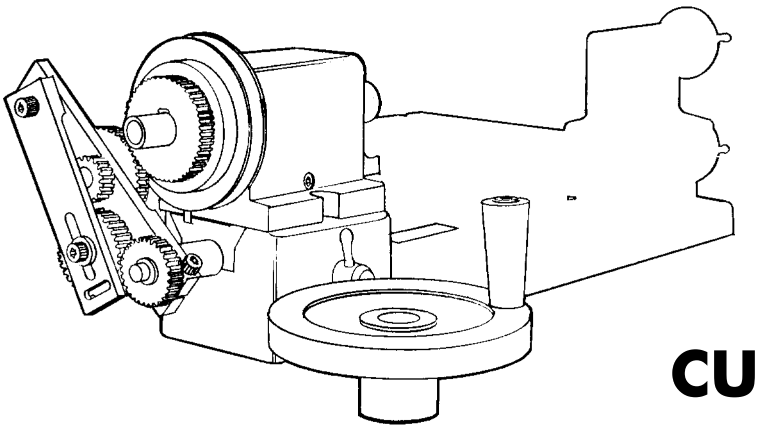


FIGURE 2— Flat area for set screw on commercial end mills.

The end mills are held with a set screw that goes into the flat ground on the shanks of all commercial end mills. (See Figure 2.) Another advantage is 3/8" end mills are available with cutters on each end of the shank creating further savings.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone: +43-2236-892 666 fax: +43-2236-892666-18

UNITurn & UNIMill
The cool tool



3" LATHE THREAD CUTTING ATTACHMENT

P/N 3100

AN INTRODUCTION TO THREAD CUTTING IN THE REAL WORLD

After designing and putting the enclosed screw cutting attachment into production, we sat down and started reading what other people have written about cutting screw threads before writing our own instructions. It amazed us that we had been able to cut threads all these years knowing so little. How and why we were able to do this is going to be the subject of our instructions. There are sufficient books available at any library to go into additional detail on the subject if required. These instructions are based on using sharp pointed 60° tools and cutting threads for your own use.

The reason other books go into such great detail on the precise methods used commercially is that they are telling you how to cut threads from specifications for other people. They have to have exact methods and standards to make sure that a bolt made in California will screw into a nut manufactured in New York. Fortunately, we have the tremendous advantage of having both pieces at hand and we can just "keep cutting until they fit". It's as simple as that! You simply select the proper gear from the chart; put in a 60° threading tool and have at it.

A point to ponder about thread cutting is how a lathe produces a thread. It doesn't matter whether the lathe is a 20" or a 3". The principle is the same. The lead screw that drives the saddle is geared directly to the spindle. When the spindle turns, the saddle moves. If they were geared one to one, the pitch cut would be the same as the pitch of the lead screw. On the 3" lathe, this would be 20 Threads Per Inch (TPI). If we turned the lead screw 180° while we turned the spindle 360° (by using a 20 tooth to a 40 tooth gear arrangement) we would cut 40 TPI. Please note that we did not have to consider the stock's diameter. The only requirement is that the major diameter is at least twice the depth of the thread plus enough material to support these threads while cutting them. One gets used to hearing a diameter called out with the threads, such as 1/4" - 20, 6 - 32, 10 - 24, etcetra, but while it's unusual to think of 40 threads per inch cut on something 2" in diameter. Yet, in some cases it may be entirely practicable to do so.

It may interest you to know how a metric thread can be cut on a 3" lathe that has American National screw threads on its lead screw. The 127 T conversion gear does this by driving the lead screw at a ratio that converts 20 TPI to 1mm. Consider 100T on the spindle driving a 127T. The ratio is .7874 to 1. The lead screw has 20 TPI: .050" P x .7874 = .03937" = 1 mm.

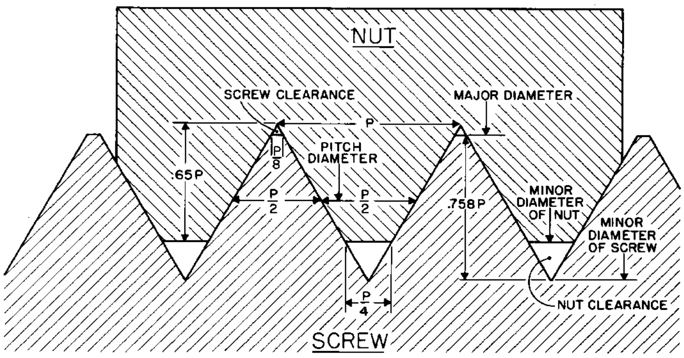


Figure 1—Component parts of a thread cut with a sharp pointed 60° vee tool.

MAJOR DIAMETER - Largest diameter of the thread of either the screw or the nut.

MINOR DIAMETER - Smallest diameter of the thread of either the screw or the nut.

PITCH DIAMETER - The theoretical diameter that falls on a point where the thread width and the groove width are the same.

PITCH (P) - The distance from point to point measured parallel to the axis. Metric threads are always expressed in Pitch

LEAD - The distance a screw thread advances axially in one turn. On a double lead screw, the lead is twice the pitch.

NOTE: The same methods can be used in figuring dimensions for American or Metric screw threads.

1 mm = .03937"

Pitch (Metric) x .03937" x .758 = depth of screw thread in inches

Take the time to familiarize yourself with component parts of the screw thread from Figure 1. The pitch diameter is the important one to consider. Before going on, let's take the time to really understand why. The pitch diameter determines

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone: +43-2236-892 666 fax: +43-2236-892666-18

how a screw or thread will fit, not the major diameter. Suppose you were cutting 20 TPI and the major diameter was .010 undersize and the pitch diameter was correct. About the only thing wrong would be that the flat on the point of the thread would be a little wide, but it would still have approximately 75 % of its strength and work well.

Now let us suppose we cut the pitch diameter undersize by .010. We would end up with a nut that fits so loose and a thread that was so weak that we would have to scrap it. There is where "cutting to fit" comes in. You can compensate for some pretty bad errors on the major and minor diameters by having the pitch diameter correct. To get it correct, all you have to do is to keep trying it for size as you cut. Don't ever take the part out of the chuck to try it because it would be next to impossible to re-chuck it in exactly the same place. However, the entire chuck, along with the part, could be removed from the lathe to try it for size. Don't force anything when trying the part for size, because you might move the part slightly in the chuck, and really "screw things up".

Why have we made such a point about having the major or minor diameter wrong and still making the part work? Read on. You're probably thinking we must really be a "hacker" if we can't cut a diameter within .010. Well, the problem in many cases, is not how close you can cut to a diameter, but what the diameter should be.

Example: Your buddy just heard you bought a nice new shiny lathe complete with a screw cutting attachment, and like all good friends, immediately goes to work trying to figure out how you and your new lathe will be of some use to him. It doesn't take him long! He has a camera which he tried to repair himself last year, but lost an important part. Of course the missing part has metric threads, but that's a "snap" for a 3" lathe. A quick check with a thread gauge indicates that it has .4 mm Pitch. No problem, yet. It is an internal thread, so you will have to cut a screw to mate with it. Here's the problem: What is the major diameter? You can measure the diameter of the hole, but you can't be assured that the thread form is perfect and that this is really the minor diameter. You can only assume that it's close. Now you take this dimension and add to it twice the depth of the thread, which should give you the major diameter. To get the depth of one thread, multiply the Pitch x .6. (Note: Pitch x 1.2 + Minor Diameter = Major Diameter). Total depth of thread using a sharp pointed 60° tool = P x .65 = .036" x .65 = .023".

The constant .6 is not used to figure depth of an external thread, it is just one used to get you in the "ball park" in a situation such as this.

At least we have a fairly reliable place to start now and can probably get one cut that will work on the first try. Always keep track of the total depth cut in case it comes out undersized. At least you'll know how deep not to cut it on the second one!

The example we gave you was one of the more difficult situations you may run into, not only because you had to do the job for free to keep a friend, but also because you had very limited information from which to work.

Usually, you will be cutting both the screw and nut. This is a case where two wrongs can almost equal one right. You can rectify any error you may have had in cutting the first one by compensating for it in the mating part.

Left-hand threads can be cut as easily as right-hand threads on a lathe; the only difference being the addition of an idler gear which reverses tool movement so that it travels left to right.

It's hard to appreciate just how much money an inexpensive lathe like this, with a screw cutting attachment can save you, until you have had to have a special part made that doesn't have a standard thread size. Even though there may be taps or dies available, a left-hand 1"-32 would probably cost half as much as your entire thread cutting attachment.

What we have tried to do in these opening remarks is to show that screw-cutting is really easy, and to give you the self-confidence it takes to do any job well. Too often, good craftsmen are stopped from venturing forth because the only information available shows the technically perfect way to do things rather than the simple, practical methods everyone really uses.

THREAD CUTTING CONVERSION KIT

This kit has been engineered to add additional versatility to your 3" Lathe. With this attachment, a wide variety of threads, both right-handed and left-handed may be produced. Most American Standard and Metric threads may be cut with equal ease and precision. The accompanying charts list the entire range from which you may choose. *(See Figure 5.)*

CONVERSION INSTRUCTIONS (Refer also to illustrations)

STEP 1. Carefully drive furnished small sheet metal screw into hole located in spindle which extends from the left side of the drive pulleys. Use a proper size screwdriver for this operation and avoid install ing the screw at an angle since it must seat squarely against the spindle. After driving, re move the screw and dress down the "burr" which will be raised around the edge of the hole. A small fine file is suitable for this. Next slide two thin spacer washers over the tube and against the pulley. Reinsert the sheet metal screw and tighten firmly.

STEP 2. Remove the headstock and loosen the flat head socket screw a few turns.

STEP 3. Remove cap screw under base and directly below headstock.

STEP 4. Grease shaft with flats on both ends and slide shaft into the lead screw support (situated directly below pulley). Be sure end with small flat enters first. Now slide shaft with a single flat into the lead screw support. To guarantee that the shaft is "home", turn shaft one or two revolutions while applying gentle inward pressure to the end of the shaft.

STEP 5. Replace screw from STEP 3 making sure that

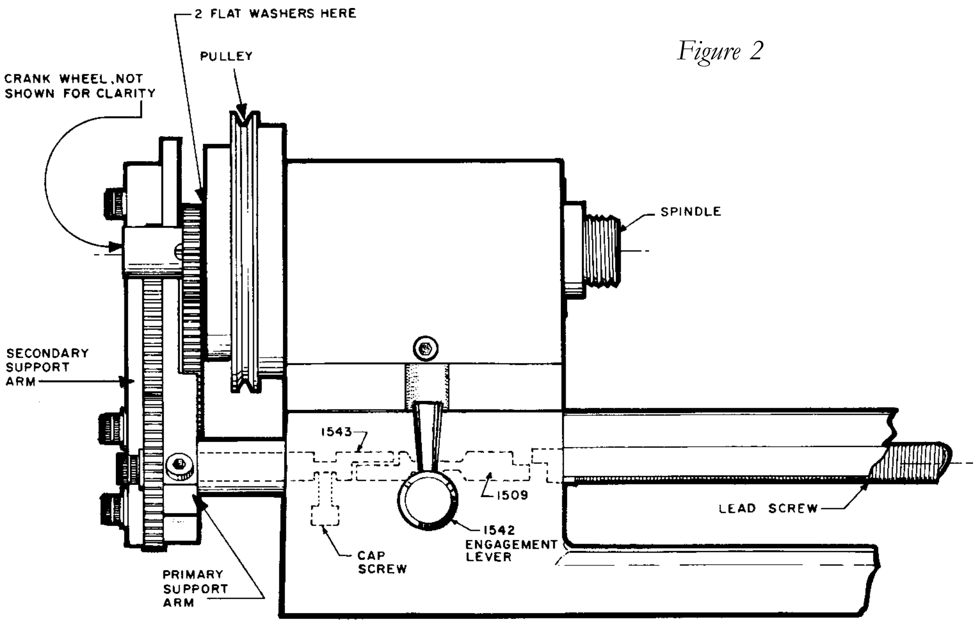


Figure 2

SIDE VIEW, THREAD CUTTING ATTACHMENT INSTALLED

point of screw goes into machined groove. Check that shaft is free to rotate. Retighten the flat head socket screw and replace the headstock.

STEP 6. Pull out the black plug below the name plate and slide the remaining shaft (with handle) into the hole (handle upward). It may be necessary to rotate the shaft about 30° backwards and forwards to get it completely "home".

NOTE: If insertion or movement of the Engagement Lever is difficult, try loosening the two screws on the bottom of the machine that hold the bed to the base. Move the bed slightly until a good fit occurs.

STEP 7. It may be necessary to deburr parts for smooth peration.

NOTE: The section below entitled "Cutting A Thread for Practice" uses the example of cutting a 28 pitch right hand thread on a 1/4" diameter piece of stock. The following numbers are based on that setup.

STEP 8. Refer to chart (Figure 5) and select type of thread to be cut. As an example, we have chosen American Standard, 28 TPI, right hand lead.

Figure 3

Setup for cutting 28 Threads Per Inch

GEAR	A	B	C	D	E
TEETH	100	100	20	28	40

NOTE: Idler Gear "E" is used for Right Hand Threads, Idler Gears "F" and "G" are used for Left Hand Threads and are, therefore, not used in this example.

Remove motor assembly (see OPERATING INSTRUCTIONS STEP 2).

Slide gear "A" (100) onto spindle engaging slot with previously installed sheet metal screw head.

Install "B" gear (100) and "C" gear (20) onto primary support arm. The drive pin is used not only to drive the "C" gear, but also to hold the "B" gear on the arm.

Install "E" gear (40) on secondary support arm.

Slide lower split end of primary support arm over the lead screw support. Adjust until "B" gear meshes properly with "A" gear (100). When mesh is satisfactory, tighten clamp screw.

Install "D" gear (28) and secure with Allen head screw and small washer. NOTE: This screw need only be finger tight and should not be used when it interferes with the secondary support arm. Adjust secondary support arm and gear for proper engagement with mating gears. When satisfactory, tighten retaining screw and pivot screw.

Install crank wheel by sliding over spindle.

Line slot up with protruding sheet metal screw head and tighten down crank wheel set screw using Allen wrench. A few drops of oil on moving parts will be helpful.

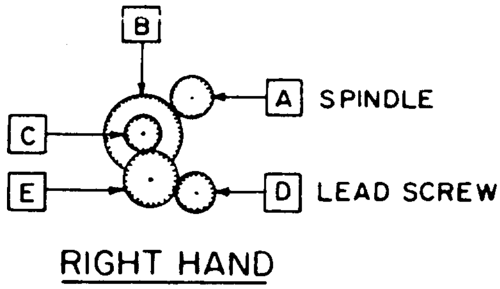


Figure 4 Gear Setup Diagram for Example

CUTTING A THREAD FOR PRACTICE

We believe the time has come to "HAVE AT IT" and start by chucking up a piece of aluminum and turning it to 1/4" diameter. Let's cut 28 TPI on it. Be sure to have a nut to check it with. Looking at the chart we see we need an "A" 100T on the spindle, driving a "B" 100T, which is attached to the "C" 20T, driving the lead screw "D" 28T, using the idler "E" 40T that mounts on the swing arm. The gears should mesh so they run "free" and have a reasonable amount of backlash. **NOTE:** All gear trains have some "backlash" and it will not effect the quality of the thread, but it does have to be allowed for. This is why the tool has to be backed out before the lathe spindle is reversed.

Over 90% of the threads cut on a lathe of this type will have a pitch less the .070, and be less than 3/8" long. Now and then you may have to cut a fairly course thread (more than .070" pitch) and a good idea is to "rough out" the thread by moving the tool post slightly to the left between passes. This keeps the tool from having to cut on both sides. On a standard lathe, the tool is advanced by the compound rest which is set at 29°. This allows only one side of the tool to cut and lessens the load considerably. The final cut is then taken with the crosslide being advanced to "clean up" the thread. We can get the same effect by moving the tool post. When cutting fine threads you can get away with cutting "straight in". The crank drive gives you the "feel" and a precise method of stopping needed in single-pointing fine threads. Cranking the spindle counter clock wise gives you reverse. This allows you to cut the entire thread without disengaging the lead screw.

Establish the depth of the first cut by bringing the tool in to the point where it just touches the surface. Write the dial setting down, and move the tool past the starting point of the thread. Now engage the lead screw lever. The lead screw may have to be turned while applying slight pressure on the lever in order to get it to engage properly.

DO NOT DISENGAGE UNTIL THE THREAD HAS BEEN COMPLETELY CUT.

Now advance tool .003" for first cut. Turn the spindle counter clockwise until the desired length of thread has been cut. Back the tool out until it is completely clear of the part. Crank spindle clockwise until tool is at the original starting point. Advance the tool to its last point plus .002". We've always found it useful to write these dial settings down too, it is amazing how fast you can forget one! Now take the second pass by cranking the spindle counter clockwise. The amount the tool should be advanced from this point on should be governed by the amount of force it took the last pass. The cut will get progressively heavier each time the tool is advanced. Remember, you can't ruin your part by taking too light a cut. To figure what the total amount the tool should be advanced if you are using a sharp Vee form tool (standard form of tool used in single pointing threads) simply multiply the pitch times .758.

Example: Pitch of 28 TPI = 1/28

Pointed tool depth = P x .758 = 1/28 x .758 = .027

If you are not too good with math and don't like to do it, just keep cutting and looking at the flat on the top of the thread. When the flat is 1/8 the pitch, the nut should fit. Either way, check it long before you think it is finished to be on the safe side until more experience is gained. The last two passes should be repeats of previous dial settings to clean up threads. Not too hard was it? No matter what type of threads you may cut, the basic method will remain the same. Internal threads are very seldom cut full depth. To figure the hole size you should start with: take the pitch of thread you are cutting and multiply it by 1.083 and subtract this from the major diameter. To figure the total depth using a sharp pointed 60° tool, multiply the pitch by .65.

EXAMPLE: To cut an internal 1-1/2"-28 TPI:

Major Diameter = 1.5

P = 1/28 = .036"

Major Diameter - (P x 1.083) = Hole Size

1.500" - (.036" x 1.083) = Hole Size

1.500" - .039 = 1.461

Hole size = 1.461

A double lead could be cut by picking change gears that are one-half the pitch and indexing the "A" gear 180° after cutting the first thread to depth.

NOTE: There isn't any way to check a double lead until it is completely cut, therefore, the depth must be figured mathematically. It has always been fun for us to do jobs like this, not because the were needed, but just to see if we could!

SCREW CUTTING OPERATION

(Read detailed instructions before proceeding.)

STEP 1. Turn or bore stock to proper diameter.

STEP 2. Remove the motor assembly from the lathe by unscrewing the two socket head cap screws that hold the motor bracket to the headstock.

STEP 3. Install thread cutting tool in post holder.

STEP 4. Place tool bit at starting point of thread and set for .003" cut.

STEP 5. Engage lever at base of lathe by turning lead screw support handle clockwise. Turn lead screw handwheel until full engagement occurs.

STEP 6. Turn spindle crank wheel until tool bit has traveled full length of intended thread.

STEP 7. Back crosslide out to clear tool from thread.

STEP 8. Turn crank wheel backwards until tool bit has traveled past starting point of thread.

STEP 9. Return crosslide to its original position plus .002".

STEP 10. Repeat STEPS 6, 7, 8, and 9 until full depth of threads has been cut. Cutting oil will make cutting easier, and will give a better finish.

FIGURE 5—GEAR SELECTION CHART FOR THREAD CUTTING ATTACHMENT

ENGLISH THREADS

THREADS PER IN.	GEAR A	GEAR B	GEAR C	GEAR D	GEAR E	GEAR F	GEAR G
80	50	100	20	40	38	28	22
76	50	100	20	38	40	30	22
72	50	100	20	36	40	28	34
68	50	100	20	34	40	28	30
64	50	100	20	32	40	28	30
60	50	100	20	30	40	28	26
56	50	100	20	28	40	26	30
52	50	100	20	26	40	24	34
48	50	100	20	24	40	26	30
44	50	100	20	22	40	26	30
40	100	100	20	40	38	28	22
38	100	100	20	38	40	30	22
36	100	100	20	36	40	28	34
34	100	100	20	34	40	28	30
32	100	100	20	32	40	28	30
30	100	100	20	30	40	28	26
28	100	100	20	28	40	26	30
26	100	100	20	26	40	24	30
24	100	100	20	24	40	26	30
22	100	100	20	22	40	26	30
20	100	100	20	20	40	26	24
19R	100	100	40	38	30		
19L	100	50	20	38		30	22
18R	100	100	40	36	30		
18L	100	50	20	36		28	34
17R	100	100	40	34	30		
17L	100	50	20	34		28	30
16R	100	100	40	32	30		
16L	100	50	20	32		28	30
15R	100	100	40	30	32		
15L	100	50	20	30		28	26
14R	100	100	40	28	30		
14L	100	50	20	28		26	30
13R	100	100	40	26	30		
13L	100	50	20	26		24	30
12R	100	100	40	24	30		
12L	100	50	20	24		26	30
11R	100	100	40	22	30		
11L	100	50	20	22		26	30
10R	100	100	40	20	30		
10L	100	50	20	20		26	24

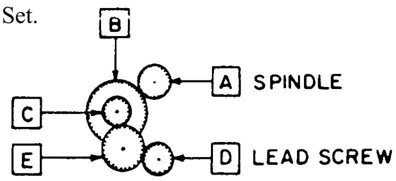
NOTE

When cutting right hand threads, Gear "E" is used in the vertical slot of the Secondary Support Arm, Part Number 3103. When cutting left hand threads, Gear "F is used in the vertical slot and Gear "G" is used in the horizontal slot and Gear "E" is not used.

METRIC THREADS

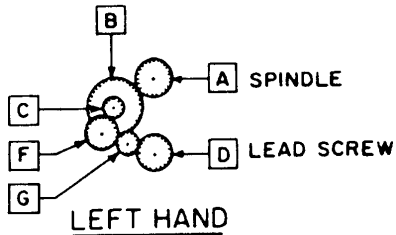
PITCH (mm)	GEAR A	GEAR B	GEAR C	GEAR D	GEAR E	GEAR F	GEAR G
.25	50	127	20	40	30	28	22
.3	50	127	24	40	30	26	22
.35	50	127	28	40	30	26	22
.4	50	127	32	40	30	24	22
.45	50	127	36	40	30	20	22
.5	100	127	20	40	30	28	22
.55	100	127	22	40	30	28	20
.6	100	127	24	40	30	28	22
.65	100	127	26	40	30	28	22
.7	100	127	28	40	30	26	22
.75	100	127	30	40	28	24	22
.8	100	127	32	40	30	24	22
.85	100	127	34	40	30	20	22
.9	100	127	36	40	30	20	22
1.R	50	127	40	20	30		
1.L	100	127	20	20		26	24
1.1	100	127	22	20	40	24	26
1.2	100	127	24	20	40	22	26
1.25	100	127	30	24	38	22	26
1.3	100	127	26	20	40	22	24
1.4	100	127	28	20	38	22	24
1.5	100	127	30	20	38	20	26
1.6	100	127	32	20	38	20	26
1.7	100	127	34	20	38	20	22
1.75	100	127	35*	20	38	20	22
1.8	100	127	36	20	38		
1.9	100	127	38	20	36		
2.0	100	127	40	20	30		

* Not included in Standard Set.



RIGHT HAND

NOTE: Gear "E" or "F" and "G" are idler gears and are used to transmit power and control direction of rotation only.

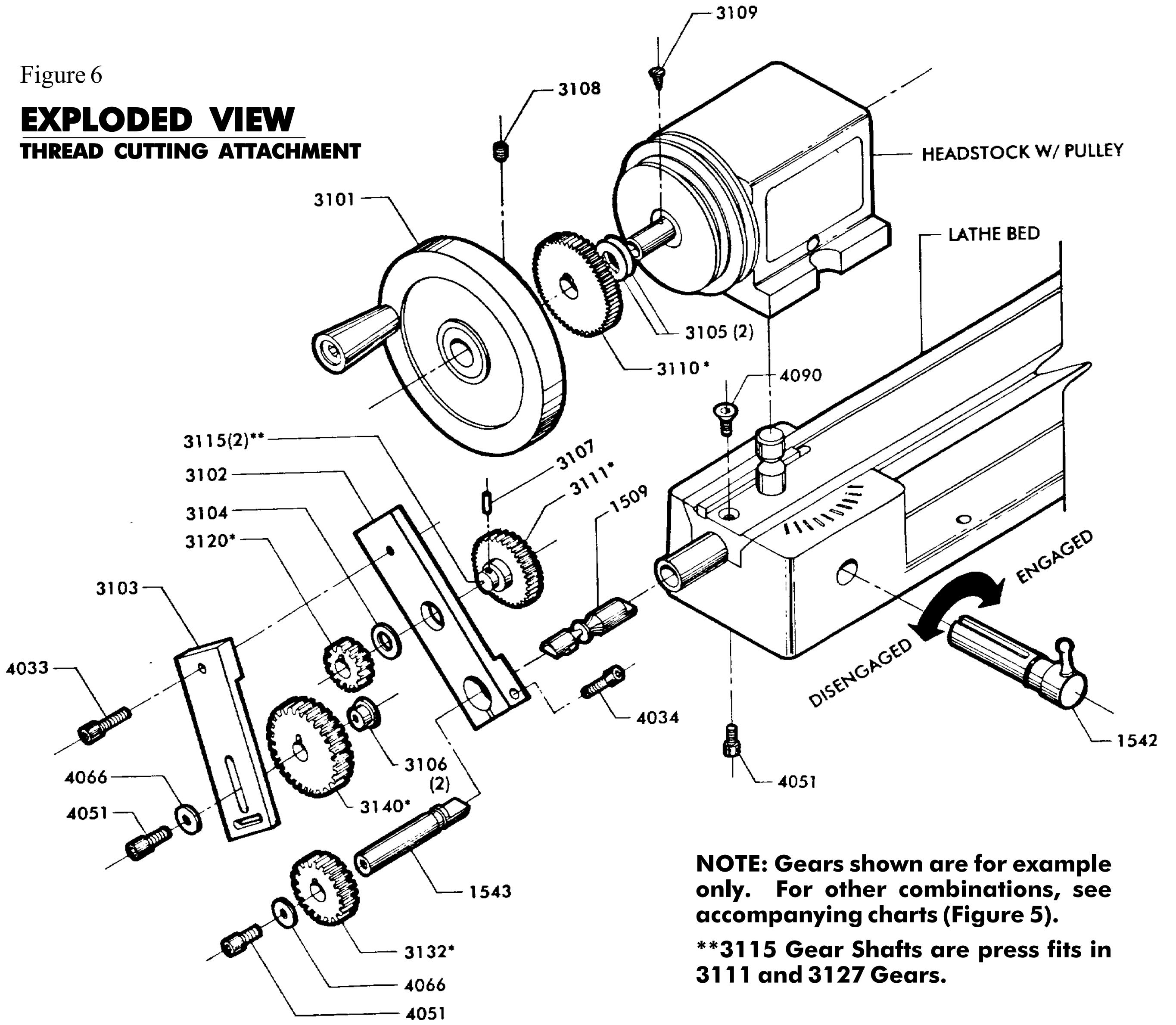


LEFT HAND

To use this chart with the model 4100 (Metric) Lathe, use the 100 tooth gear in place of the 127 tooth gear when cutting metric threads and the 127 tooth gear in place of the 100 tooth ("A" Gear) when cutting American threads. Press the shaft out of the 127 tooth gear and into the 100 tooth gear. American threads finer than 40 TPI cannot be cut.

Figure 6

EXPLODED VIEW
THREAD CUTTING ATTACHMENT

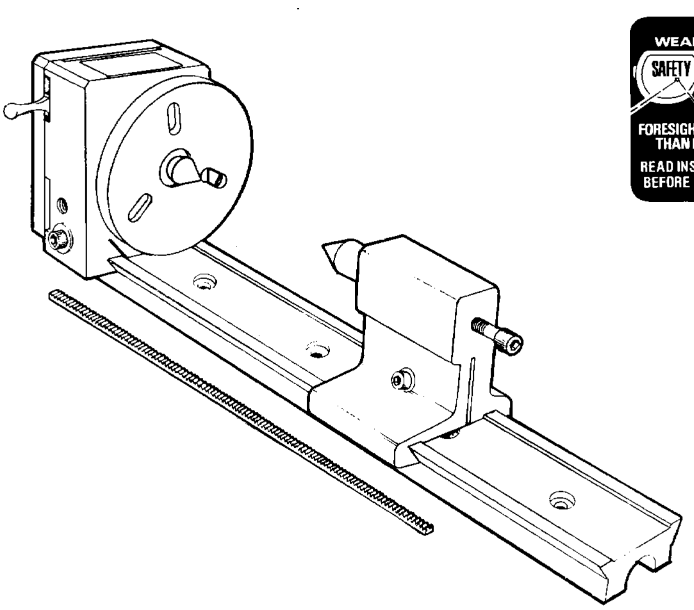


NOTE: Gears shown are for example only. For other combinations, see accompanying charts (Figure 5).

****3115 Gear Shafts are press fits in 3111 and 3127 Gears.**

PARTS LIST, THREAD CUTTING ATTACHMENT

PART NO.	DESCRIPTION	PART NO.	DESCRIPTION
3101	HANDWHEEL	3120	20 TOOTH GEAR, 24 PITCH
3102	PRIMARY SUPPORT ARM	3122	22 TOOTH GEAR, 24 PITCH
3103	SECONDARY SUPPORT ARM	3124	24 TOOTH GEAR, 24 PITCH
3104	SMALL SHIM WASHER	3126	26 TOOTH GEAR, 24 PITCH
3105	LARGE SHIM WASHER (2)	3127	127 TOOTH GEAR, 56 PITCH
3106	GEAR BUSHING (2)	3128	28 TOOTH GEAR, 24 PITCH
3107	GEAR DRIVE PIN	3130	30 TOOTH GEAR, 24 PITCH
3108	10/32 x 3/8" SET SCREW	3132	32 TOOTH GEAR, 24 PITCH
3109	SHEET METAL SCREW, PAN HEAD, NO. 6 x 3/16", TYPE A	3134	34 TOOTH GEAR, 24 PITCH
		3136	36 TOOTH GEAR, 24 PITCH
3110	100 TOOTH GEAR, 56 PITCH (w/ notch)	3138	38 TOOTH GEAR, 24 PITCH
3111	100 TOOTH GEAR, 56 PITCH	3140	40 TOOTH GEAR, 24 PITCH
4034	10-32 x 1" SKT HD CAP SCREW	3150	50 TOOTH GEAR, 56 PITCH
4051	10-32 x 3/8" SKT HD CAP SCREW (3)	1509	SLIDING SHAFT
4033	10-32 x 5/8" SKT HD CAP SCREW	1542	ENGAGEMENT LEVER
4066	NO. 10 WASHER	1543	FIXED SHAFT
3115	GEAR SHAFT (2)		



UNITurn & UNIMill
The cool tool

INDEXING ATTACHMENT

P/N 3200

OPERATING INSTRUCTIONS

This Indexing Attachment has been designed to give the average hobbyist an all-purpose method of dividing circles into an equal number of segments to aid in cutting gears or any other repetitive, circular machining operation. It is of a price and size which makes it ideal for use with miniature machines. The dividing head can be used in both horizontal and vertical modes.

Although it has been designed to be used with the , it can be adapted for use with other types of equipment or used for different purposes described in this booklet.

Before attempting any machining operation, be sure your set-ups use good machining principles and practices. Work in a careful, professional, craftsmanlike manner, and **ALWAYS WEAR SAFETY GLASSES**.

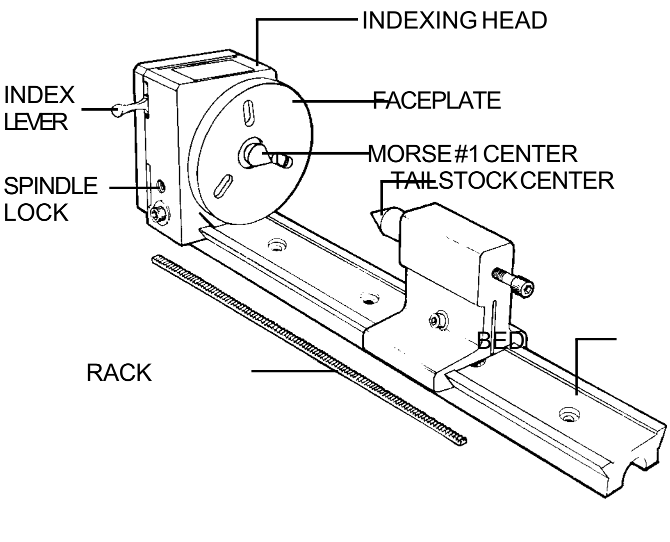


FIGURE 1--Parts of the Indexing Attachment

LUBRICATION AND MAINTENANCE

Like any fine machine tool accessory, care should be taken to keep your indexing attachment clean and free from rust. Moving parts should be oiled periodically with sewing machine oil. The indexing head can be easily taken apart for cleaning when necessary.

ADJUSTMENTS

End play can be removed from the head spindle by unlocking set screw #3214 (Ref #16 on exploded view) and turning clockwise to remove “play”. Turning the set screw counter-clockwise reduces drag on the spindle.

TWO METHODS OF DIVIDING

1. INDEXED METHOD. This method is quite simple and uses the indexing lever and the graduated scale on the spindle. Internally the indexing lever engages with a 72-tooth gear, and each tooth equals 5° of movement. Obviously this method will only allow indexing of simple hole patterns since you can only work in multiples of 5°; however, this is usually sufficient for most jobs with the exception of cutting gears. Since very few gears will work out in even multiples of 5°, a second method of dividing called the “calculated method” can be used. It is described below.

It is important to remember to lock the spindle before attempting any machining. The indexing lever is NOT a lock and is not intended for any use other than to locate the index (Seee Figure 2).

2. CALCULATED METHOD. This method will yield an infinite number of divisions but takes considerably more time. To set up the head in this mode, the indexing lever must be raised to its uppermost position. The rack gear is then inserted from either side with the teeth towards the spindle under the lever. It is important that the spindle lock is loose so the spindle is free to move as the rack is inserted. The theory behind the calculated method should be apparent now. As the spindle is rotated, the rack moves in a linear motion which can be easily measured. If the total move-

The cool tool®
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18
www.thecooltool.com

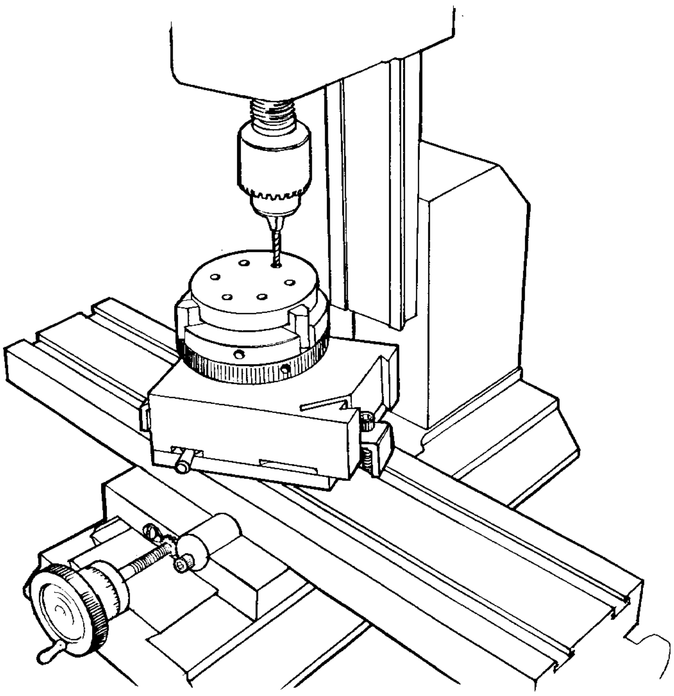


FIGURE 2--INDEXED METHOD, Drilling a precise hole pattern.

ment of the rack for one revolution is known, any number of divisions can be made by dividing this dimension by the number of divisions required.

The calculated linear dimension for one complete revolution is 4.712 inches (119.685mm), but this dimension may vary slightly from one indexing attachment to the next. For utmost precision, it is suggested that you accurately measure your particular indexing head and note the dimension for future use. Use a precise vernier or dial caliper of at least 5 inches in length (6 inches is preferable) equipped with a depth rod (See Figure 3).

To determine this dimension for your particular indexing head, drill a small hole with a center drill on the very top edge of the faceplate with the indexing head mounted on its bed. Before drilling be sure the rack is positioned in such a way that one complete revolution can be turned and still have the rack properly engaged. Make sure to lock the spindle and accurately measure from the end of the rack to the indexing head. After drilling, unlock the spindle and rotate it one revolution using the hole and center drill to index the spindle. Measure the rack again and subtract the smaller number from the larger. The difference should come out quite close to 4.712" or 119.685mm. Since the accuracy of all your machining is dependent on the precision of this measurement, it is suggested that you DOUBLE CHECK your work!

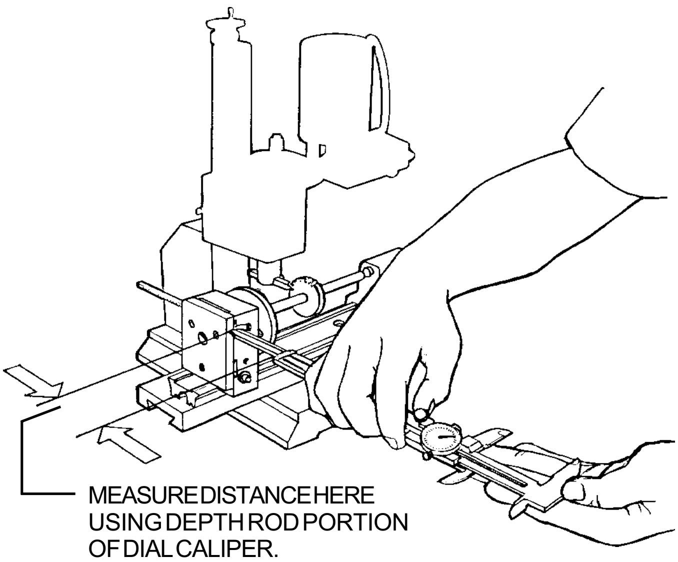


FIGURE 3-- Calculated Method, measuring the rack position.

Once you have this dimension, dividing a part into any number of divisions is easy. Just divide 4.712" (or the dimension for your attachment if different) by the number of divisions you wish to make. This will give you the distance the rack must move for each division. The math is simple, but a small pocket calculator saves a lot of time.

EXAMPLE:

Say you want to cut an 83-tooth gear, 56 pitch (The size of the blank can be arrived at from the Machinist's Handbook and a similar type gear can be used to arrive at the cutter shape). The blank can be mounted on an arbor and held between centers. The dog is clamped on the arbor in such a way that it engages with the faceplate. Care must be taken to eliminate all "play" where the dog engages with the faceplate. Grind a tool bit that will give the desired tooth shape and use this tool like a flycutter. Cutters can be purchased that will generate an accurate shape, but the are very expensive and hard to find. With a little practice excellent results can be obtained with the "flycutter method" (See figure 4).

Once the cutter is properly held in a holder and has been located on center using the tailstock center for a reference point, the indexing attachment is properly clamped to the machine table and aligned with an indicator, and the rack gear is located in such a way a complete revolution can be turned, you are ready to begin.

MAKING THE FIRST CUT

Before you begin "making chips", look again at your set-up; is it really SAFE? Also make sure you're wearing safety glasses. Remember, that cuts of this type require very rigid set-ups because of the intermittent cutting action. If the gear blank is thin it may require additional support by sandwiching the gear blank between support pieces shaped like a large washer and of a material which is easily machined.

Turn on the machine spindle and move the "Y" axis toward the cutter while moving the "X" axis back and forth until the cutter just starts to touch the blank. Write down the dial setting and calculate the total depth of the cut. (The information to calculate this can be found also in the Machinist's Handbook.) The first cut should be less than .010" (.20mm) deep. Observe the cutting action carefully. Is the cutter

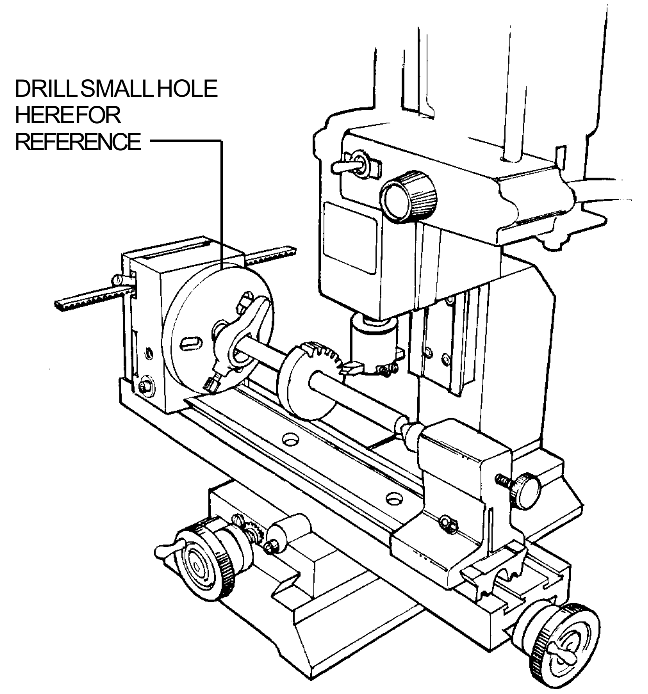


FIGURE 4-- Typical set-up for cutting gear teeth.

cutting properly? Is there excessive vibration in the set-up? Is the cutting speed proper? There is no book written that can give you the answer to these questions; this is where experience and craftsmanship come into play. The best way to make good parts is to work **VERY CAREFULLY!** To cut an 83-tooth gear means you have to do 83 successive machining operations correctly to make a good part...82 out of 83 is a waste of time!

Once your cutting speed and feed are to a point you're sure you can repeat the same operation over and over again with excellent results, finish out your first cut to its final depth. Now it is time to index for the next cut. Measure the distance from the end of the rack to the index head carefully before unlocking the spindle. Write down this dimension (it will be referred to as "A"). From previous instructions you have already figured the total throw of your indexing head—say it is 4.712". You now divide this number by the number of divisions, in this case 83 to get: 4.712 x 1/83 = .056771", or rounding off, .057". This is now added to or subtracted from dimension "A". At first glance it would appear that all you need to do is add .057" for each cut because an error of only .000229" is so small it can be discounted. But if you multiply this error by the total number of teeth (in this case 83) you would end up with an error of .019" which would make the last tooth you cut a very "interesting" shape. This is what is known as "tolerance buildup" and is the reason you must use your basic formula at each step to calculate the next dimension rather than simply adding rounded off dimensions. The second cut and each succeeding cut are calculated as follows: 4.712 x 2/83=.113542 or .114". Add this to or subtract from dimension "A", index and cut.

With each cut you make your understanding of the techniques involved will increase. By working and thinking in a careful manner you should be successful on the first attempt.

Although the instructions given here have been related to cutting a gear, the same approach must be used for any type of indexed machining.

FIVE BASIC RULES TO REMEMBER

1. Work ACCURATELY.
2. Determine the best possible way of holding the part to be machined. This type of machining requires a very secure set-up.
3. Carefully align the set-up with the machine.
4. Take cuts that the set-up you are using can easily withstand.
5. Don't try to rush the job! Successive machining operations make some people lax; therefore, it is wise to consider the amount of time and effort you will lose if you destroy your part rather than how much is left to do.

INDEXING ATTACHMENT SPECIFICATIONS

OVERALL LENGTH	12 INCHES
DISTANCE BETWEEN CENTERS	7 INCHES
MAX. DIAMETER HORIZONTALLY	3.50 INCH.
GRADUATIONS	5 DEGREES
SPINDLE THREAD	3/4-16

INDEXING ATTACHMENT
EXPLODED VIEW

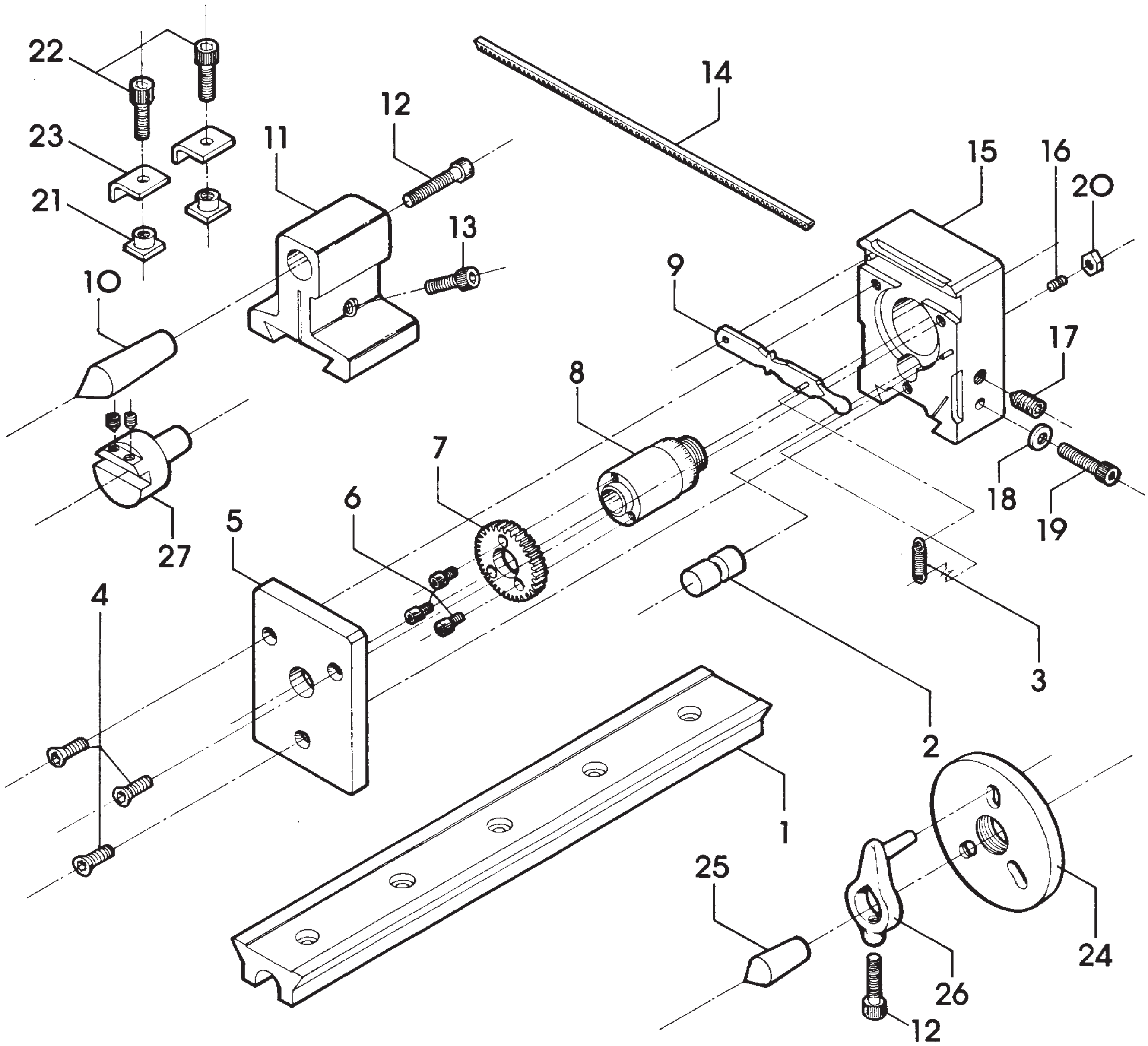
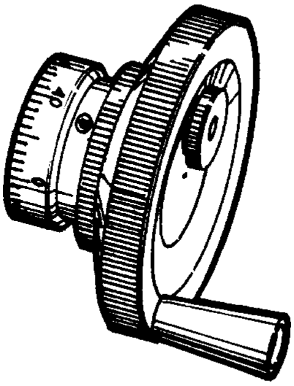


FIGURE 5

REF #	PART #	DESCRIPTION	REF #	PART #	DESCRIPTION
1	3201	BED	15	3213	INDEXING CASE
2	3202	LOCKING PIN	16	3214	SKT HD SET SCREW, CUP PT, 10-32 X 1/2"
3	3203	SPRING	17	4054	SKT HD SET SCREW, CONE PT, 5/16-18 X 3/4"
4	3219	SKT HD FLAT SCREW, 10-32 X 1//2"	18	3114	#10 S.A.E. WASHER
5	3220	INDEXING CASE COVER	19	3215	SKT HD CAP SCREW, 10-24 X 1"
6	3221	SKT HD CAP SCREW, 6-32 x 3/8"	20	3216	10-32 HEX NUT
7	3222	INDEXING GEAR, 72 TOOTH, 48 PITCH	21	3056	TEE NUT, 10-32
8	3223	SPINDLE	22	4033	SKT HD CAP SCREW, 10-32 X 5/8"
9	3224	STEPPING LEVER	23	3558	HOLD DOWN CLAMP
10	3225	TAILSTOCK CENTER	24	4007	FACEPLATE
11	3226	TAILSTOCK CASE	25	4038	MORSE #1 CENTER
12	4034	SKT HD CAP SCREW, 10-32 x 1"	26	4009	DRIVE DOG
13	4050	SKT HD CAP SCREW, 10-24 x 7/8"	27	3217	GEAR TOOTH CUTTER HOLDER
14	3212	RACK, 48 PITCH	28	3218	INSTRUCTION MANUAL



UNITurn & UNIMill
The cool tool

RESETTABLE HANDWHEELS

PART NUMBERS: 2" – 3420 (Inch), 3430 (Metric)
2½" – 3440 (Inch), 3450 (Metric)
2½" Assembly, "Z" Axis – 3455 (Inch), 3459 (Metric)

Most expensive full size machine tools allow the machinist to reset the handwheel to "zero" (or any desired setting) at any time during a machining operation. Now that option is available on UNITurn's miniature machine tools as well.

INSTALLATION

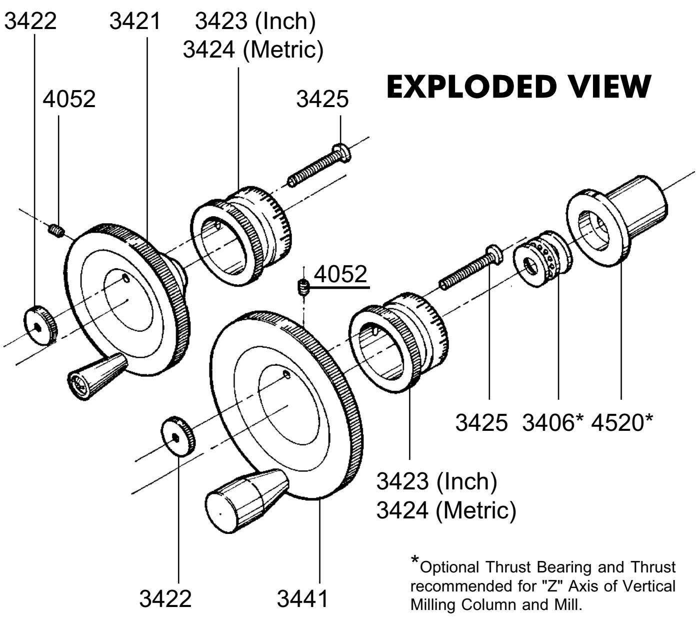
The resettable handwheels easily replace any standard UNITurn handwheel. Simply loosen the set screw on the standard handwheel and slide it off the shaft. Slide the new handwheel onto the shaft. Align the hole in the engraved collar with the set screw. Eliminate all "play" (backlash) between the handwheel and column thrust and tighten the set screw.

The larger 2½" handwheel is normally used on the "Z" axis of the Mill or Vertical Milling Column and works best when used with the thrust and bearing, because you are actually "lifting" the weight of the column with this handwheel when you crank it up. Handwheels turning on the horizontal axis are not subjected to this stress and work fine without thrust bearings. Later model UNIMill DeLuxe Mills and Vertical milling Columns include the "Z" axis thrust bearing as standard. If you are upgrading an older Mill or Column that does not have a thrust bearing on the "Z" axis, you will need to order P/N 3460 which consists of a handwheel, a bearing set and bored column thrust. If your existing handwheel has a thrust bearing in it, you will need to purchase P/N 3470 which includes a new resettable handwheel and a bored column thrust. (You will use the ball bearing set from your old handwheel.) Simply remove the bearings from your old handwheel, set them into the new bored column thrust and install it in place of the old column thrust.

RESETTING THE HANDWHEEL

At any time during your machining operation, you can now simplify your calculations by resetting the handwheel to "zero". To do so, gently hold the handwheel in position with one hand while releasing the lock nut with the other. Rotate the red anodized, laser engraved collar until the "zero" setting is aligned with the scribed mark on the Mill, Lathe or thrust bearing collar. Then retighten the locking nut. Now you can crank in the exact amount of feed you want by reading the number directly off the handwheel.

EXPLODED VIEW

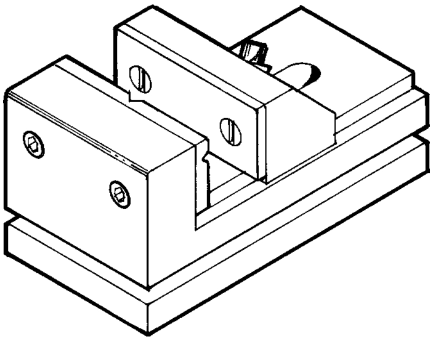


*Optional Thrust Bearing and Thrust recommended for "Z" Axis of Vertical Milling Column and Mill.

PARTS LIST

PART #	DESCRIPTION
3406	Thrust Bearing and Washers
3420	2" Handwheel Assembly, Inch (Metric P/N 3430)
3421	2" Handwheel body
3422	Handwheel Locking Nut
3423	Engraved Hndwhl Collar, Inch (Metric P/N 3424)
3425	6-32 x 7/8" Pan Head Screw
3426	"Z" Axis Hndwhl Collar, Inch (Metric P/N 3427)
3440	2½" Handwheel Assembly, Inch (Metric P/N 3450)
3441	2½" Handwheel Body
3455	2½" Hndwhl Asby, "Z" Axis, Inch, (Met. P/N 3459)
4052	10-32 x 3/16" Cup Point Set Screw
4520	Bored Column Thrust
3460	2½" Hndwhl w/Thrust & Bearings (Met. P/N 3465)
3470	2½" Hndwhl w/ Bored Thrust (Metric P/N 3475)

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18



UNITurn & UNIMill
The cool tool

MILL VISE
P/N 3551

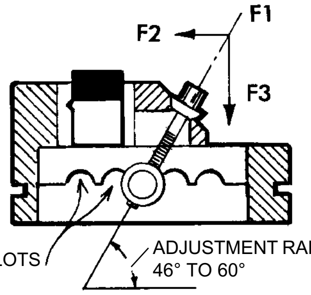
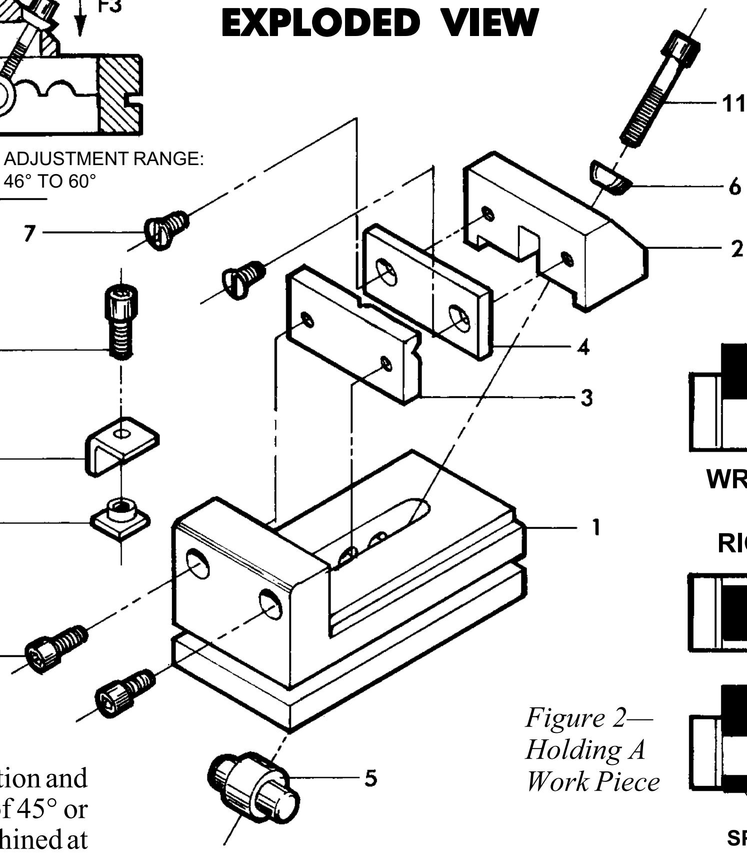


Figure 1—
Pull-Down Feature

EXPLODED VIEW



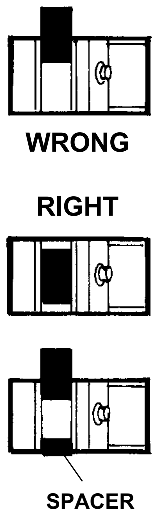


Figure 2—
Holding A
Work Piece

INSTRUCTIONS

The advantages of this vise are obvious when movement of the jaw is studied. (See Figure 1 and look at the bottom of your vice.) The tightening force (F1) produces not only a force against the part (F2), but also a force pulling the jaw downward (F3). Therefore, angle "A" must exceed 45° in order to make force F3 greater than F2. This keeps the movable jaw from "tipping" back. Also note that extreme clamping angles beyond 60° start to apply much downward pressure but not much horizontal force is directed to holding the part. Moving the pull-down barrel to the proper slot keeps the adjustment within the most effective clamping range.

To clamp a part, place the jaw in approximate position and start tightening the adjustment screw at an angle of 45° or greater. (The back face of the moveable jaw is machined at a 45° angle for reference.) If the angle of the adjustment screw gets up to 60° or greater and you still haven't drawn down on the part, loosen up the screw a little and move the pull-down barrel to the next slot and retighten.

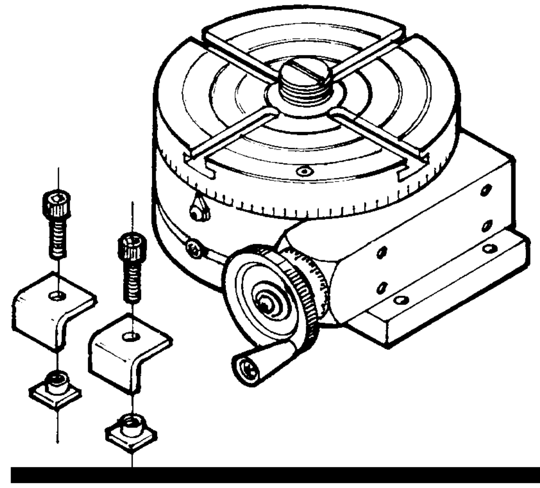
***CAUTION!** Extreme vertical adjustment angles can allow the 10-32 x 1-3/4" adjustment screw to be driven into the surface of the table, damaging both screw and table. Therefore, the vice comes assembled with a 1-5/8" screw which is usable for most settings. To enhance adjustment to the longest ranges, change to the 1-3/4" screw provided.*

Figure 2 shows the proper way to hold a part in the vise. If the part cannot be centered, use a spacer to help keep the jaws parallel. This vise has been designed to acurately hold objects being machined. It is not recommended for use as a bench vise or for clamping parts in such a way and with such force as to adversely affect its accuracy.

MILL VISE PARTS LIST

REF NO.	PART NO.	DESCRIPTION
1	3511	Vise Body
2	3502	Movable Jaw
3	3503	Fixed Jaw Insert
4	3504	Movable Jaw Insert
5	3512	Pull-Down Bar
6	3506	Convex Washer
7	3507	Flat Head Screw, 6-32 x 3/8 (2)
8	3056	T-Nut, 10-32 (2)
9	3558	Hold Down Clamp (2)
10	4033	Skt Hd Cap Screw, 10-32 x 5/8" (4)
11	3513	Skt Hd Cap Screw, 10-32 x 1-5/8"
12	4070	Skt Hd Cap Screw, 10-32 x 1-3/4"

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone: +43-2236-892 666 fax: +43-2236-892666-18

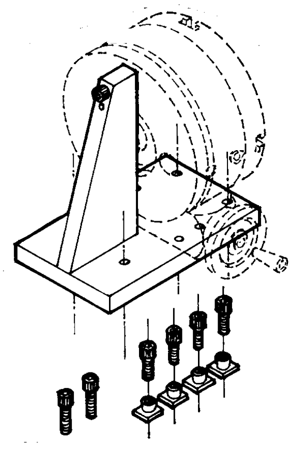


UNITurn & UNIMill
The cool tool

ROTARY TABLE
P/N 3700

UNIMill DeLuxe Products' rotary table is 4" (100mm) in diameter and has been designed to be used in conjunction with their vertical mills; however, it can be easily adapted to any equipment where size and configuration would make it useful. It has a worm ratio of 72-1 making one revolution of the handwheel 5° of table movement. The table has been engraved with 5° lines identified every 15°, and the handwheel has 50 graduations making each graduation 1/10° allowing a circle to be divided into 3600 parts without interpolating. The table can be locked by tightening set screw ref. #24 of the exploded view.

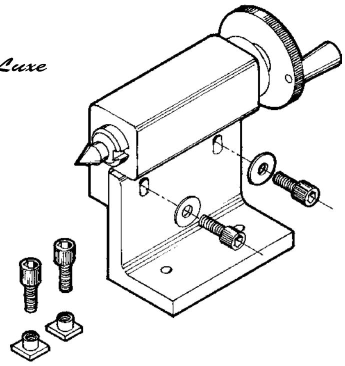
The T-slots accept UNIMill DeLuxe 10-32 T-nuts (P/N 3056 or 4025). The weight of the rotary table is 7 pounds and it stands 2" (50mm) high; it has been built of bar stock steel.



Optional UNIMill DeLuxe right angle attachment (P/N 3701).

A right angle attachment (P/N 3701) is available. This has been designed with an adjustment to align the table perfectly vertical. (See separate instructions at end of rotary table instructions.)

An adjustable right angle tailstock (P/N 3702) is also available to allow you to turn a part between centers using the rotary table, right angle attachment, and adjustable right angle tailstock. (See instructions on page 6 at end of rotary table instructions.)



Optional UNIMill DeLuxe adjustable right angle tailstock (P/N 3702) allows for accurate turning between centers when the optional right angle attachment (P/N 3701) is used.

The following instructions have been written to show what's involved in doing a complex job accurately. We believe if you truly understand the job we will describe in detail, average jobs will be accomplished without filling your trash can with mistakes. Remember, there are not many people capable of making the complex machined products used today, and if you can master the vertical mill and the rotary table combination, you will have come a long way at becoming a good machinist. You will find erasers aren't much good and no one has come up with a good "putting on" tool when it comes to metal parts. Complex parts are very difficult to make. When you're making "one-of-a-kind" parts, don't worry how long it takes; spend your time planning and checking so you don't have to worry about starting over.

When a rotary table is put on a vertical mill you end up with a machine that is theoretically capable of reproducing itself. This means the capabilities of your UNIMill DeLuxe are governed by the size of the part and the ingenuity of the operator. The purpose of these instructions is to give you an insight into properly using this accessory. An inexpensive calculator with trig functions is a must for complex jobs.

Standard milling machine setups usually involve aligning the work with the table and then with the spindle. This is easily accomplished because the table can be accurately

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone: +43-2236-892 666 fax: +43-2236-892666-18

moved with the handwheels. Aligning a part on a rotary table can be very trying because the work has to be clamped into position. When you consider the fact that the part turns, a .001" (.03mm) error in location gives a .002" True Indicated Reading (T.I.R.) run-out when checked with a dial indicator.

Many times it is advisable to start by doing the rotary table work first which can eliminate precision aligning. A quick way to align the milling spindle with the rotary table is by indicating the hole in the center of the rotary table. Next, prick punch or spot drill the center on the work you wish to have line up with the rotary table. Put a pointer in the spindle that runs true. Set the work under the spindle and lower the head until it engages with the center mark, then clamp the part down. You now have the work reasonably aligned with the rotary table and spindle. At this time, rotate the table with the spindle running and the pointer slightly backed off. If the part is properly aligned, the pointer should always line up with the center mark, and you should write down your handwheel settings. It is also advisable to write an "R" or "L" after the handwheel setting to remember which way the backlash was set.

Enclosed with your rotary table is an adapter (P/N 3709) that allows a UNIMill DeLuxe chuck to be screwed directly to the table. This allows work that is of the correct size and configuration to be quickly aligned with the rotary table with reasonable accuracy. Be sure to consider the fact that a mill cutter could unscrew a 3- or 4-jaw chuck held on in this fashion (See Figure 1). Use only very light cuts when this

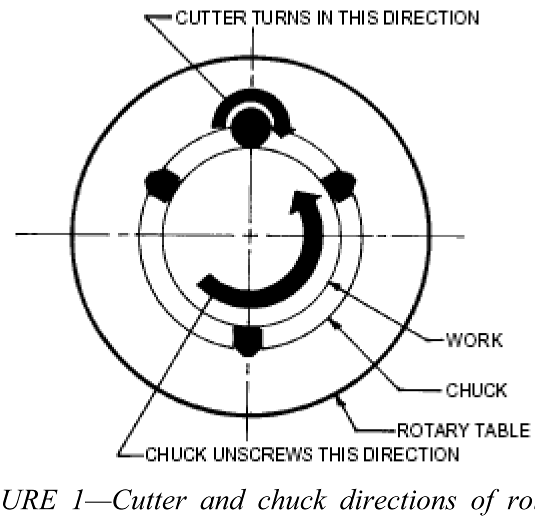


FIGURE 1—Cutter and chuck directions of rotation.

adapter is used. If you believe this could be a problem with your set-up, add a second clamp to eliminate the possibility of the chuck unscrewing from the adapter.

The ball game changes when you want perfection and this is true whether you are working with an inexpensive UNIMill DeLuxe tool or a $20,000 mill. You can't expect to work within .001" unless you have your machine square.

On the UNIMill DeLuxe a few shims and a dial indicator should get your machine square if you have something square to work to, preferably a small precision square. There is no adjustment for "X"-axis in relation to "Y", but this has been machined accurately. The vertical slide should be square with the table and the head and spindle should be square with the vertical slide. Remember that the size of the part has a lot to do with how square the machine has to be.

The first place to start to align your UNIMill DeLuxe is to run an indicator on the work table to check for flatness. Move the X- and Y-axes independently to determine any error. These errors can be easily eliminated by placing a shim under the rotary table so the table runs perfectly true. Normally, this isn't necessary, but we are talking about "perfection".

To align the vertical bed with the "X" and "Y" slide, clamp something to the table that you are sure is square. With an indicator mounted to the head, move the head up and down a couple of inches with the indicator reading a known square that is set up to read in the "X"-axis direction. With the four screws that hold the steel bed to the column block, adjust the bed until there is a minimum indicator movement. The "Y"-axis direction can be corrected with a shim between the column block and the mill base using the same method.

With the vertical bed aligned with the base, the head can bealigned to the rest of the machine by "sweeping" the head in. The rotary table will give a good surface to indicate in. Clamp the indicator in the spindle as shown in the mill instructions that came with your mill. The head should be fairly square but can be improved upon by using the slight amount of play on the alignment key to square it up on the "X"-axis and a shim between the head and saddle (if needed) on the "Y"-axis.

In most cases the job can usually be done without going through the process outlined and using the machine as it comes. I'm only trying to educate you to what it takes to work at a precision level of machining. Any toolmaker worth his salt would not attempt to build a close tolerance part without first squaring the spindle of a vertical mill.

MAKING ALLOWANCES FOR CUTTER DIAMETER

A close look at Figure 2 will start making you aware of the complexities of working with a rotary table. Unless you are doing a hole layout, you very seldom can work with the angles and dimensions on your drawing because of the cutter diameter.

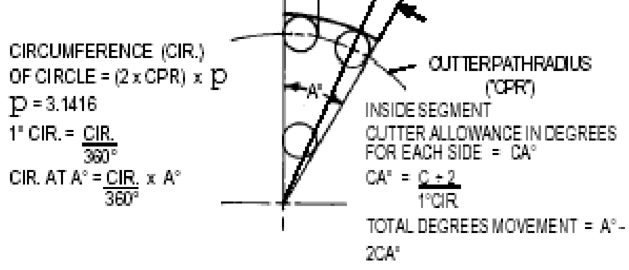


FIGURE 2—A demonstration of CPR or Cutter Path Radius.

Figures 3 and 4 show the relation of cutter and part. Start considering what I refer to as CPR, which is where the center of the cutter is from the center of the rotary table.

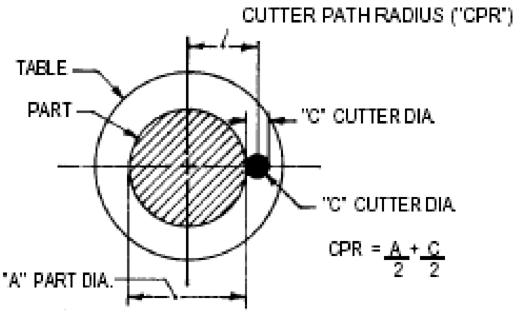


FIGURE 3—Cutter machining outside of part.

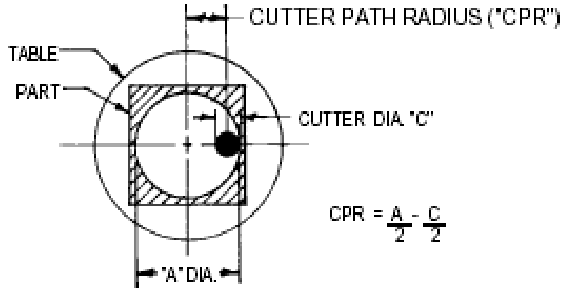
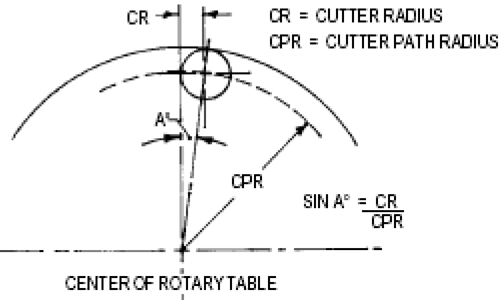


FIGURE 4—Cutter machining inside of part.



CPR = 1.375 CUTTER DIAMETER = .250

$\frac{\text{CUTTER DIAMETER}}{2}$ = CUTTER RADIUS = CR

SIN A° = $\frac{CR}{CPR} = \frac{.125}{1.375}$ = .0909

FROM TRIG. TABLE: A° = 5° 13' = 5-13/60° = 5.216°
FROM CALCULATOR: A° = 5.215°
CUTTER ALLOWANCE DEGREES = A° = 5.215°

FIGURE 5—This example shows how easy it is to allow for the cutter diameter using trigonometry.

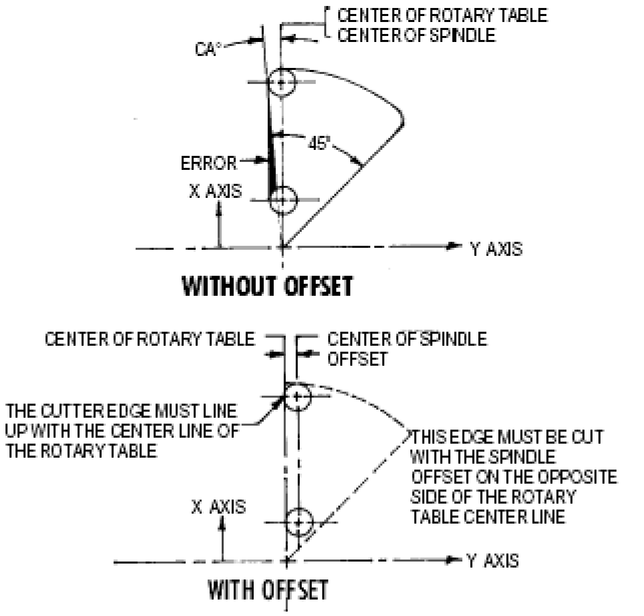


FIGURE 6—Offsetting the Rotary Table to cut segments.

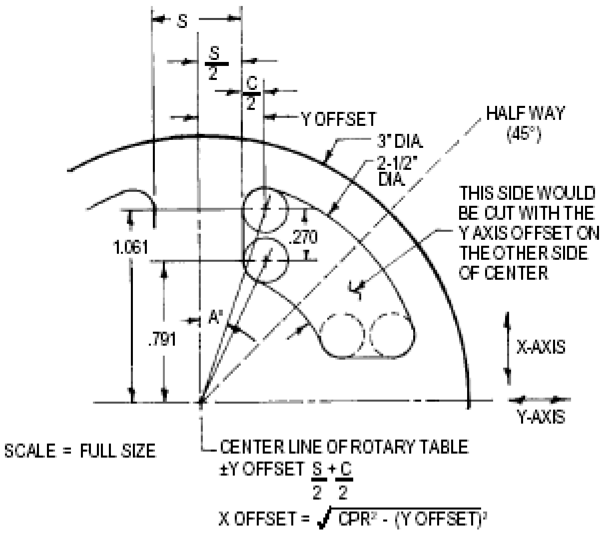
UNIMill *DeLuxe* now offers adjustable "zero" Adjustable handwheels for our lathes and mills. This makes calculation of the feed much easier as the handwheels can be reset to "zero" each time. If you do not have the resettable handwheels, the job simply requires a bit of note-taking. If you get into the habit of writing your handwheel setting down and calculating movements, it's really not bad. A piece of tape stuck along the edge of the mill table and mill base with a mark that shows starting and finishing points can be of considerable help. Of course, you will still have to use your handwheel numbers, but the marks will make you aware they are coming up. Counting the turns of a handwheel on long movements can have disastrous results if you're distracted and turn one too many. One of our customers attached scales (rulers) to our mill on both the "X" and "Y" axes which I always thought was a good idea. If you have trig tables or a calculator with trig functions you can take a lot of the guess work out of exact locations and angles.

The next problem you must be aware of is why the rotary table must be offset to cut segments. Study Figure 6 and it becomes obvious that allowing for the cutter diameter at one end of the segment will not make any correction at the other.

EXAMPLE: CUTTING A WHEEL WITH SPOKES

When one of our customers purchases their first metal cutting tool, it is usually a lathe and somewhere in that customer's mind is a brass canon he now has an opportunity to build. When a customer buys his first rotary table, chances are they either want to drill hole patterns which shouldn't require any instructions or make some kind of wheel with spokes in it. Therefore, I will describe how to "accurately" cut a wheel with spokes. I realize that in most cases it is not necessary to work to this degree of accuracy to do a job of this nature, but to accurately make a precision part of this type is what a rotary table is all about. In most cases, I will leave you to your own common sense as to the depth of cuts and how much to leave from roughing and finish cuts. Remember, I have never seen a part scrapped from taking too light of a cut.

Make an accurate drawing at the start showing offsets and cutter paths (similar to Figure 7). The offsets can be calculated as shown in the sample in Figure 7.



CUT OUTSIDE

CPR = CUTTER PATH RADIUS

CR = CUTTER RADIUS = $\frac{\text{CUTTER DIA.}}{2}$

GIVEN: S (SPOKE WIDTH) = .5

CPR = 1.250 - .125 = 1.125

C (CUTTER DIA.) = 0.25

$\pm Y\ OFFSET = \frac{S}{2} + \frac{C}{2} = \frac{.5}{2} + \frac{.250}{2} = .375$

$X\ OFFSET = \sqrt{CPR^2 - (Y\ OFFSET)^2}$

$= \sqrt{1.125^2 - .375^2} = \sqrt{1.266 - .1406} = 1.061$

CUT INSIDE

CPR = .750 (HUB RADIUS + .125 [CUTTER RAD.]) = .875

$X\ OFFSET = \sqrt{CPR^2 - Y\ OFFSET^2} = \sqrt{.875^2 - .375^2} = .791$

DISTANCE BETWEEN INSIDE & OUTSIDE OFFSETS = 1.061 - .791 = .270

FIGURE 7—Drawing and calculations for cutter paths.

REMEMBER...the rotary table center must be precisely located below the spindle when you start. Only one half ofthe segment may be cut from the calculated point which is why only one half of the spoke width is considered. Look at the drawing again and be sure you truly understand why you can only cut one half of the segment before proceeding or your chances for success will be dismal.

Now we have the offsets calculated and the rotary table "indicated in" in relation to the spindle. We move the "X" axis the amount of the offset moving the table to the left. Be sure to consider the backlash, and it may also be prudent to allow for roughing and finish cuts. Now move the Y-axis and the "Y" offset in (towards the column). This will allow the first half of the segment to be cut so that it looks like the diagram. Assuming the part is properly clamped to the rotary table and held in such a way that you can't inadvertently cut into the table, it's time to start. The example has four equal segments which means a spoke will be cut every 90°; therefore, a lot of confusion can be eliminated if you start with your table at 0° (see Figure 8). The center of the spokes will now lay out at 0°, 90°, 180°, and 270°, and the halfway point will be at 45°, 135° etc. Allowance for the cutter was taken care of when the offsets were calculated. It is not necessary to caluclate the value of angle "A" or other angles because you are only cutting one-half the segment at a time.

A good rule now is to take a very light cut (.001") and convince yourself everything is correct. The real trick of machining is to do something you have never done before the"1st time" and you can't be too careful. A one minute check versus 3 hours or more to start over makes this a good investment in time. The cut along the spoke is accomplished by moving the "X" axis only back and forth using the calculated points until you get through the part, and again we remind you it may be wise to take a roughing cut. Sometimes an undersize (resharpened) end mill is a good way to rough cut and then change end mills for finish passes. This allows the same handwheel number used for roughing and finishing.

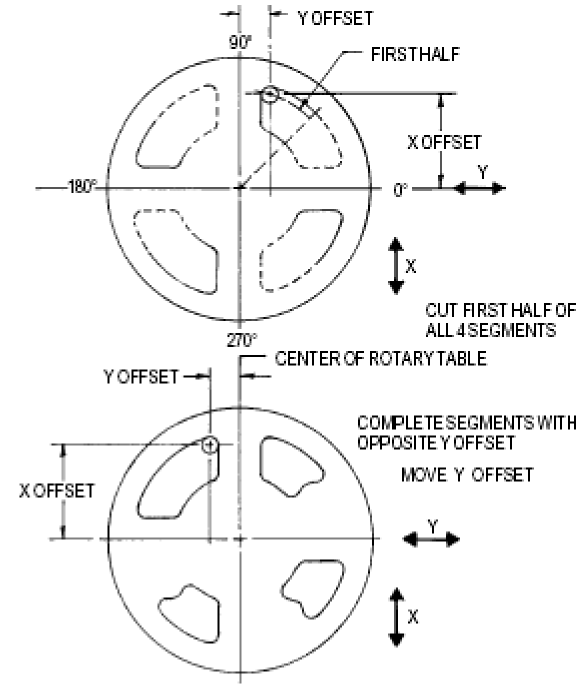


FIGURE 8—Completing the spokes of a wheel.

The rotary cuts are made with the X-axis in its proper position, and the table rotated counter clockwise. One of the real neat things in machining happens when using a rotary table to feed work into an end mill, and I believe it comes about because of the slow and precise feed that can be obtained. If a hole you're cutting requires a bottom, great finishes can be had from end mills and rotary tables. The rotary part of the segment only needs to be moved slightly past the half way point for the remainder of the segment will be cut with the Y-axis offset moved out from the column and the table rotated in a clockwise direction.

It's quicker to cut the first half of all four segments, then move the Y offset and complete the segments. If you're going to try something like this for a first project, check your entire plan out with .001" cuts and be positive you're correct before making cuts that could scrap your part (see Figure 8).

CUTTING GEARS WITH A ROTARY TABLE

I'm going to leave it up to you to determine when you know enough about gears to try and produce one. One of the best sources for information on gears is *Machinery's Handbook.* Gears are built to a rigid set of rules, and they are more complex than you might imagine.

We will only try to explain how to cut a simple, low tolerance gear. You will also have to determine the blank size, depth of cut, RPM of the spindle and so on. If you successfully cut a good gear on your first attempt, be very proud of yourself. It can be frustrating if you are not organized.

Gears can be cut using a rotary table with a reasonable amount of precision. In many cases, gears--even inexpensive ones—are very precise. Gears are usually produced by "hobbing". This method uses a cutter that is similar to a worm gear. The teeth are generated with both the cutter and the blank turning. In fact, the process looks just like a worm gear running. Methods like this produce perfectly shaped teeth that are perfectly spaced. It may be theoretically possible to produce a perfect gear one tooth at a time, but your odds of success are dismal if this is the type of gear that is required. I suggest you stick with "clock" type gears for your first few projects.

Cutters can be purchased that will produce a fairly good tooth form, but they are expensive and have a very limited range. A cutter can be ground that works like a fly cutter. Use our P/N 3217 for this. A 1/4" lathe tool blank is provided which fits this holder. Use the damaged gear you are replacing for a shape reference to grind the tip of the cutter. The corners on a bench grinder wheel are used to generate the shape on the tool blank. At first it may seem almost impossible to do this, but it is not. Just keep checking the tool to a gear that can be used for a gauge by holding the two up to a light source. You'll find that the final grinding is done by "feel". Lathe tool bits are cheap and available, so it is a process worth learning. When the tool is mounted in the holder, don't allow it to stick out any more than necessary. Figure 9 above shows a typical setup. A tailstock isn't always necessary. Remember, the gear blank must run true before starting.

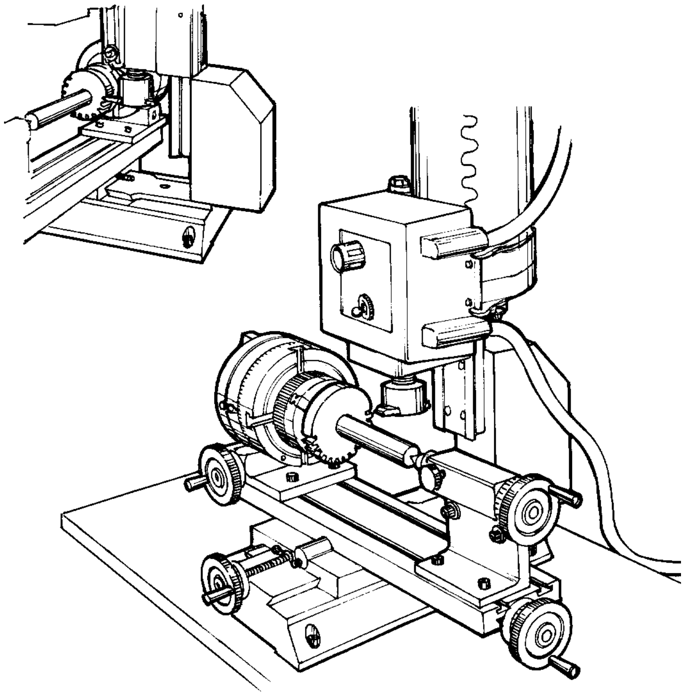


FIGURE 9—A sample setup for cutting a gear. The small inset shows the column moved back to the rear hole to allow clearance for cutting larger diameters.

CALCULATING YOUR CUTS

To figure the amount to move between cuts, an electronic pocket calculator is very helpful. Simply divide 360° by the number of teeth you wish to cut. This will give you an answer in degrees and tenths that can be used directly on your rotary table without conversion to degrees, minutes and seconds. Your rotary table is calibrated directly in degrees and decimal divisions of a degree.

EXAMPLE: CUTTING A 29-TOOTH GEAR

(Note: I have purposely used a number of teeth that does not easily divide into 360° as this will normally be the situation in which you will find yourself.)

Here are the calculations and handwheel settings you would need to cut a 29-tooth gear. Remember that the table is marked every 5° and one revolution of the handwheel is 5° which is divided into 50 parts. Therefore, each line on the handwheel equals 1/10 of a degree. Figure 10 below shows how the handwheel settings would look for the first four cuts on the 29-tooth gear:

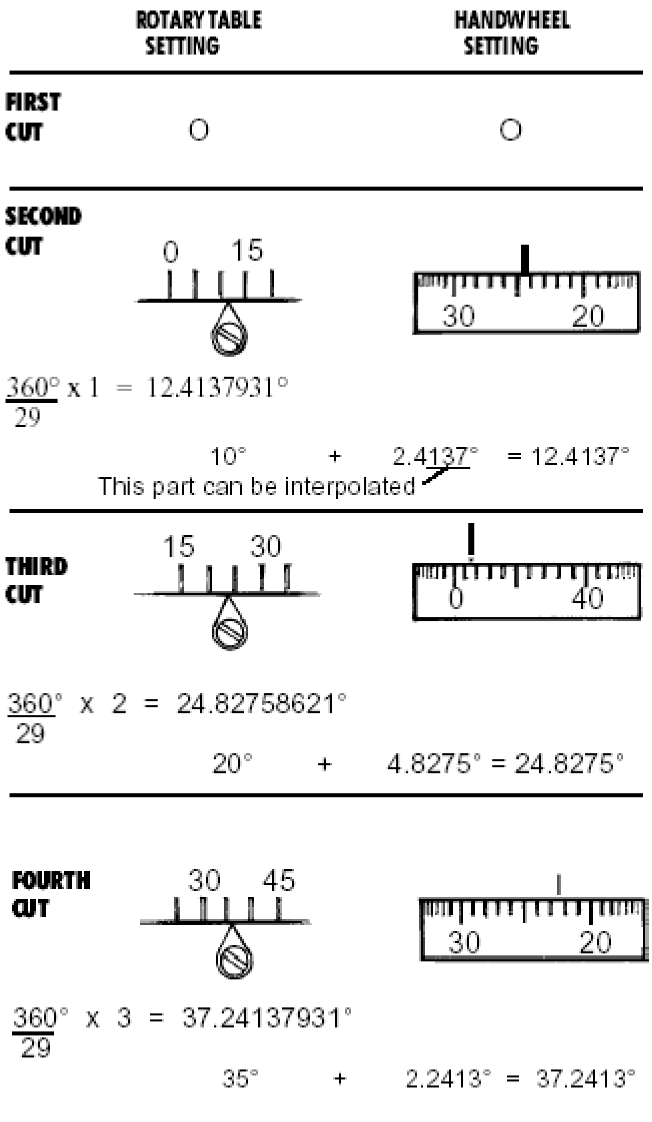


FIGURE 10—Degree calculations and handwheel settings for making the first four cuts on a 29-tooth gear.

The reason you should divide and then multiply each time is if you "rounded off" on the first division is that otherwise your error would build up by the number of teeth you were cutting. If your pocket calculator has a memory function there is an even easier method of calculating each cut. Simply store the first in memory and add it to itself each time. Because the calculator stores the number to even more decimal places than it displays on the screen, the answer is usually so accurate the 29th calculation should yield almost exactly 360°.

1. First calculation: $\frac{360°}{29}$ = 12.4137931° (2nd cut)
2. Press "Memory" key (usually "M" or "M+") to store (Remember that the first calculation is actually for the second cut, because the first cut is made with the handwheels both set at "0".)
3. Press [+] key
4. Press [recall] key
5. Press [=] key (3rd cut)
6. Press [+] key
7. Press [recall] key
8. Press [=] key (4th cut)

etc.

MAINTENANCE

Keep oiled to prevent rust. A few drops in the oiler before using will eliminate table wear. The worm gear is greased at the factory.

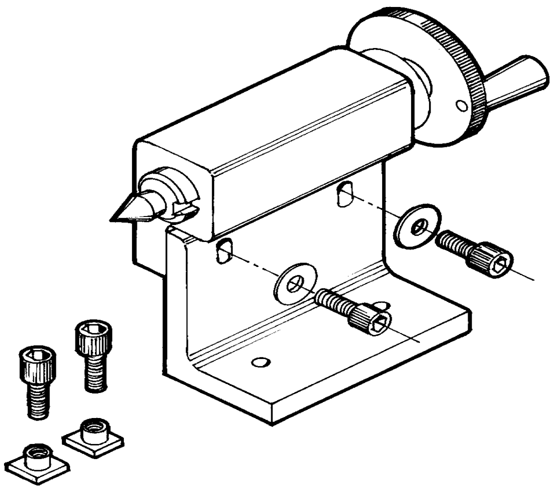
Worm backlash can be eliminated by moving the worm housing to compensate for wear.

If our instructions seem somewhat redundant, please forgive me, for I could be living on a 60' yacht in the south pacific if we could recover the money in scrapped parts we've had employees produce by cutting first and thinking later!

ADJUSTABLE RIGHT ANGLE TAILSTOCK PART NO. 3702

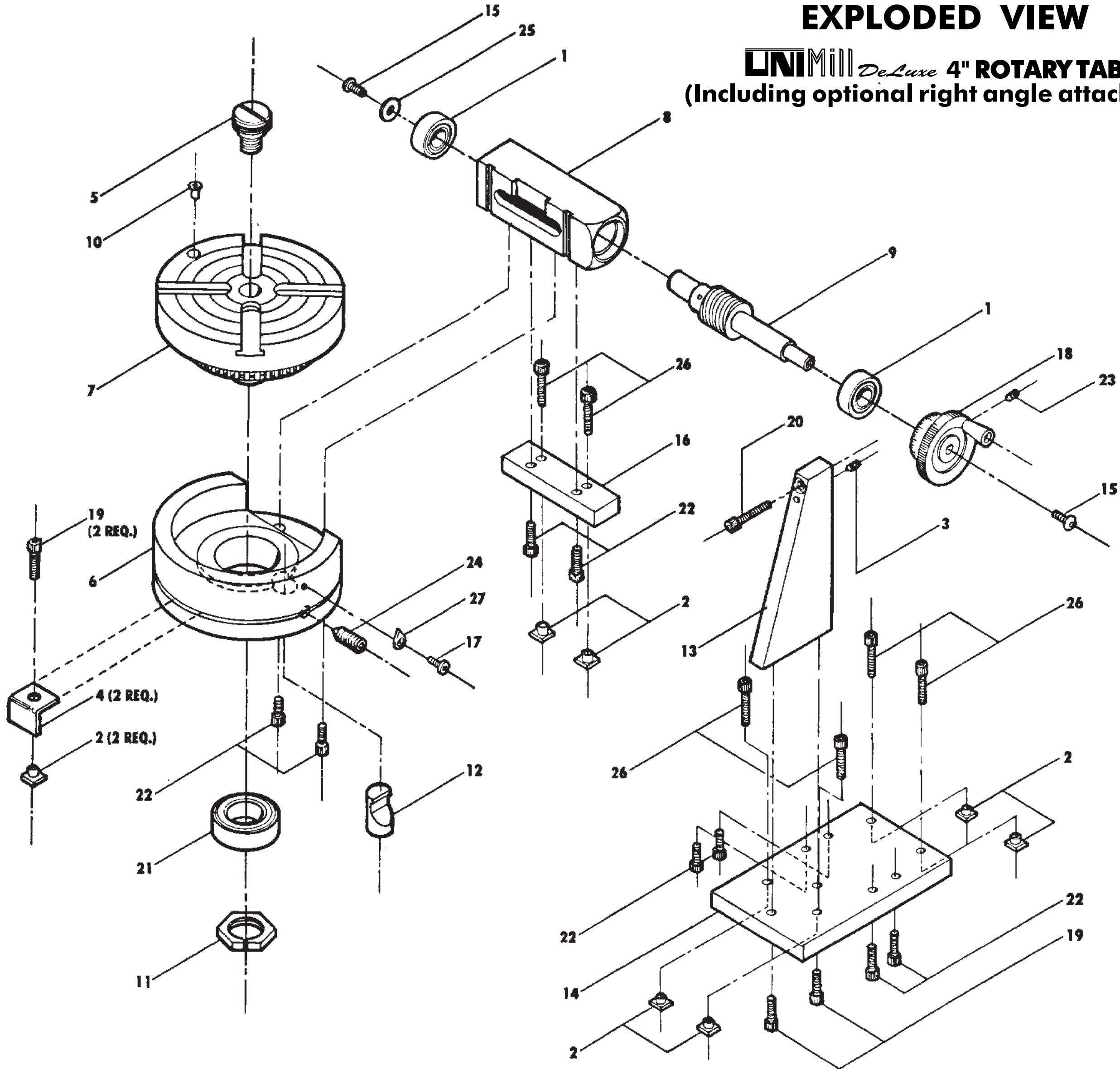
(See also Figure 9 on page 5.)

Because of tolerance build-up, it would be just about impossible to offer a tailstock that was perfectly on center with the rotary table/right angle attachment combination. The solution offered here is a modification of our standard tailstock which allows it to be adjusted to exactly line up with the center of the rotary table in order to allow for perfect alignment between the rotary table and the tailstock while holding long parts between centers.The base is attached to the mill table with cap screws and T-nuts. The two socket head cap screws go through elongated slots in the side of the right angle piece and allow for minor adjustments in height when making your setup.



EXPLODED VIEW

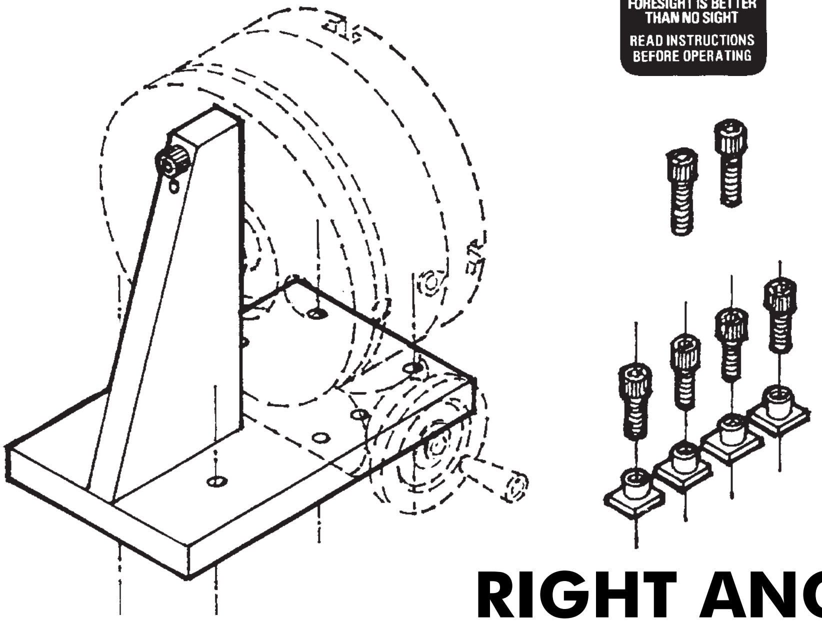
UNIMill DeLuxe 4" ROTARY TABLE
(Including optional right angle attachment)



PARTS LIST

REF#	PART#	DESCRIPTION
1	1093	3/8" Bearing
2	3056	T-nuts, 10-32 T-nuts
3	3108	Set Screw, 10-32 X 3/8"
4	3558	Hold Down Clamp
5	3709	Chuck Adaptor
6	3710	Rotary Table Base
7	3711	Table
8	3712	Worm Housing
9	3713	Worm Gear
10	3715	Oiler
11	3716	Preload Nut
12	3717	Lock Pin
13	3718	Upright
14	3719	Right Angle Base
15	3720	Button Hd Skt Hd Cap Screw, 10-32 X 3/8"
16	3721	Hold Down Tab
17	3722	Button Hd Skt Hd Cap Screw, 6-32 X 1/4"
18	4005	Handwheel Assembly
19	4067	Skt Hd Cap Screw, 10-32 X 1/2"
20	4034	Skt Hd Cap Screw, 10-32 X 1"
21	4042	Headstock Bearing
22	4051	Skt Hd Cap Screw, 10-32 X 3/8"
23	4052	Cup Point Set Screw, 10-32 X 3/16"
24	4054	Cone Pt Set Screw, 5/16-18 X 3/4"
25	4066	Washer, 3/16" I.d.
26	4067	Skt Hd Cap Screw, 10-32 X 1/2"
27	5012	Pointer

UNITurn & UNIMill
The cool tool



RIGHT ANGLE ATTACHMENT FOR 4" ROTARY TABLE

P/N 3701

The UNITurn Right Angle Attachment has been designed to easily put the Rotary Table on a vertical plane and still maintain rigidity.

INSTALLATION

Remove the hold down tab (see Rotary Table Exploded view, part #16) from the worm housing and loosely bolt the Right Angle Attachment base (part # 14) to the housing with the Rotary Table base. Back out the vertical adjustment screw (part # 3) and start the vertical clamp screw (part # 20), but do not tighten. Tighten the four Right Angle Attachment base to worm housing screws and mount to milling table with the Rotary Table indicated in with the "Y" axis. The vertical plane can be aligned by moving the indicator up and down with the "Z" axis while reading the table. The vertical clamp and set screw can now be adjusted for "0" indicator reading. The accuracy that must be attained when indicating the Rotary Table in is somewhat determined by the size of the part.

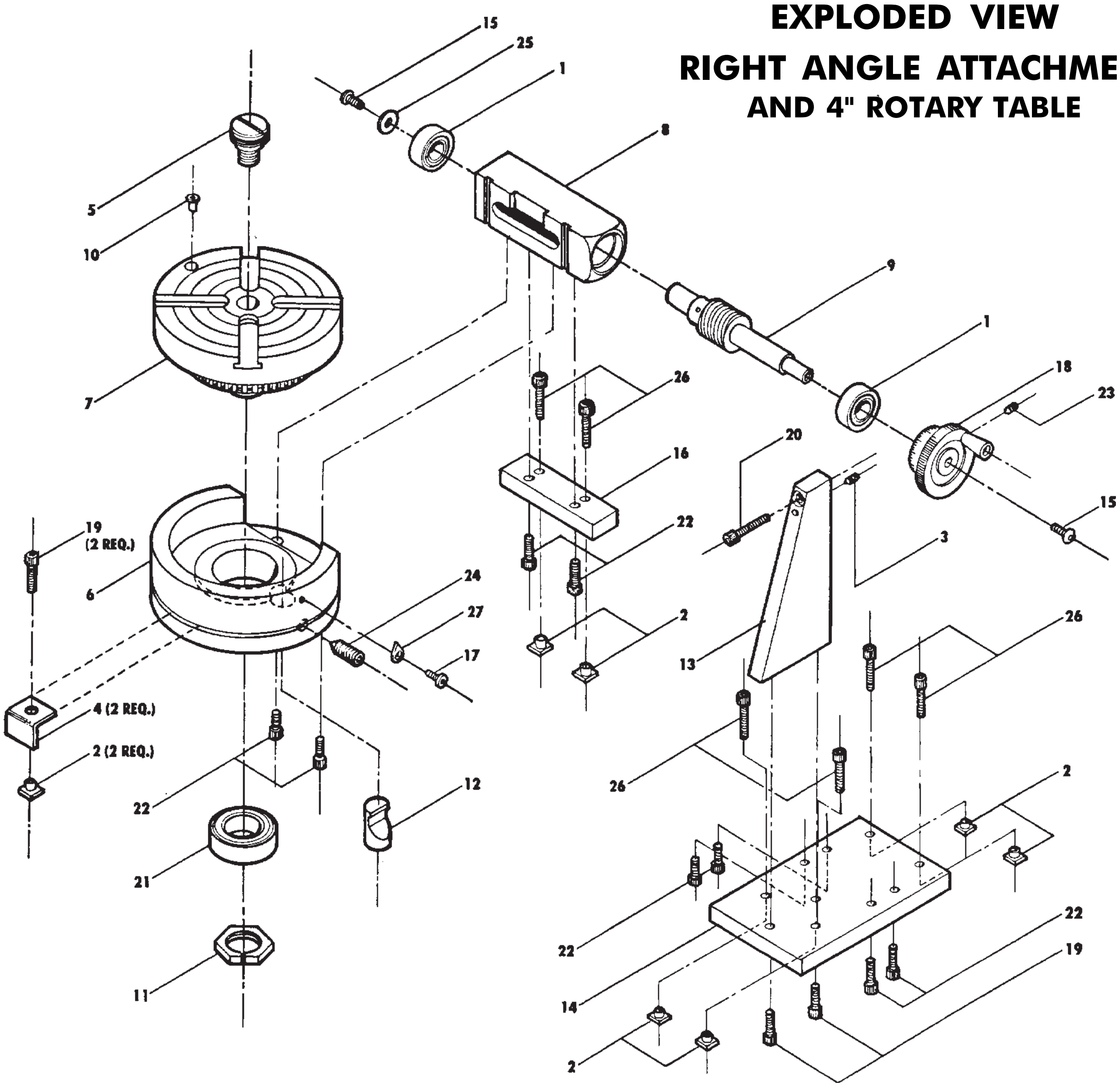
In many cases it is wise to align and clamp the part to the table before bringing the Rotary Table to the vertical position. In this manner you have the milling machine spindle to help align the part. Aligning the milling machine to the work with the Rotary Table in the vertical position is usually accomplished by measuring in from a side of the part with an edge finder or "touching off" with a cutting tool. Fortunately, you would very seldom have to align the spindle to the Rotary Table in both axes. If the need arises and you don't have a True (TIR) running surface to work to, try and leave yourself a "machining pad" on your part to do this. Once the Rotary Table has been aligned to the mill, use an end mill to machine a flat on the "machining pad" with the side of the end mill, moving the "Y" axis to determine depth of cut and "X" axis for length of cut. Rotate part 180° and cut to identical handwheel readings. Now measure across these flats and move the "Y" axis one half of this dimension plus one half the cutter diameter towards the center with the cutter out of the way. Rotate 90° and "touch off" the end of the cutter on a flat that was machined to determine center. The "Z" axis can be lowered one half the diameter to put the tools on center. If these pads are left on the work, other cutting tools could be located in the same manner and then be machined off when they are no longer needed.

You will find this accessory interesting, but difficult to use without a lot of planning.

Note: For information purposes, the exploded view drawing that accompanies this part includes all parts for the 4" Rotary Table as well, although they are **not included with the Right Angle Attachement when ordered by itself**.

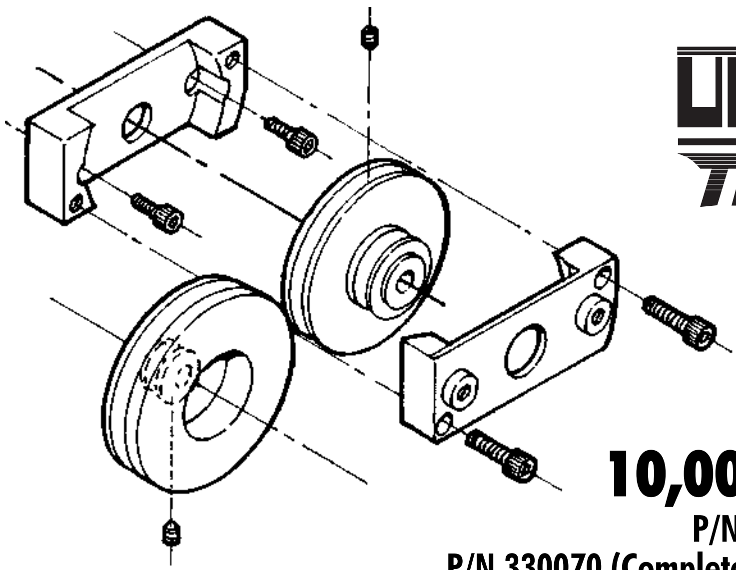
The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

EXPLODED VIEW
RIGHT ANGLE ATTACHMENT
AND 4" ROTARY TABLE



PARTS LIST

REF#	PART#	DESCRIPTION
1	1093	3/8" BEARING
2	3056	T-NUTS, 10-32 T-NUTS
3	3108	SET SCREW, 10-32 x 3/8"
4	3558	HOLD DOWN CLAMP
5	3709	CHUCK ADAPTOR
6	3710	ROTARY TABLE BASE
7	3711	TABLE
8	3712	WORM HOUSING
9	3713	WORM GEAR
10	3715	OILER
11	3716	PRELOAD NUT
12	3717	LOCK PIN
13	3718	UPRIGHT
14	3719	RIGHT ANGLE BASE
15	3720	BUTTON HD SKT HD CAP SCREW 10-32 x 3/8"
16	3721	HOLD DOWN TAB
17	3722	BUTTON HD SKT HD CAP SCREW, 6-32 x 1/4"
18	4005	HANDWHEEL ASSEMBLY
19	4067	SKT HD CAP SCREW, 10-32 x 1/2"
20	4034	SKT HD CAP SCREW, 10-32 x 1"
21	4042	HEADSTOCK BEARING
22	4051	SKT HD CAP SCREW, 10-32 x 3/8"
23	4052	CUP POINT SET SCREW, 10-32 x 3/16"
24	4054	CONE PT SET SCREW, 5/16-18 x 3/4"
25	4066	WASHER, 3/16" I.D.
26	4067	SKT HD CAP SCREW, 10-32 x 1/2"
27	5012	POINTER



UNITurn & UNIMill
The cool tool

10,000 RPM Pulley Set

P/N 4335 (Pulley set only for retrofit)
P/N 330070 (Complete Headstock/motor/speed control)

Purpose of the high speed pulley set

Over the years we've had several requests for a spindle that would turn at higher RPM. The fact that we use a 6,000-RPM DC motor made this a difficult task to accomplish and maintain our present motor mounting hardware. The main purpose of the high-speed spindle is for turning small diameters on the lathe or for turning small diameter cutters at a higher RPM, which makes them less prone to breakage. We do not consider this an accessory to use like a standard accessory. You have to take the drive system apart to add it, and it will take about ten minutes to change over. There is a second belt position; however, you also have to take the drive system apart to change from high-speed to low-speed. This is a design compromise needed to allow you to fine-tune your Sherline machine while still maintaining a reasonable cost.

Safety considerations

You have to realize that when you are dealing with high-speed spindles you are dealing with a different "animal." You have to consider the safety of your setup before you turn the spindle on. An unsupported shaft turned at high RPM may suddenly wobble and bend 90º. Having a part come loose because of a poor set up and dance around your workbench while still spinning at 10,000-RPM is dangerous. Both ends of the spindle must be considered because the work that may be sticking out the back end of the spindle will be unsupported. As with all machining operations safety glasses are a must. Do not spin large diameters or out of balance setups at high RPM on a Sherline. Remember, these are lightweight machines and cannot tolerate errors of this type.

Adjusting preload for high speed operation

At the factory we set the endplay of the spindle at 0.0002". We have found that this setting is too "tight" for use at this high an RPM and increased it to 0.0003" for spindles that are going to run continually for long periods. If you are mounting the high-speed accessory on a spindle that may have had considerable use, you may not have to change the adjustment. In any case I'd run it and check to see if it gets too hot before making any adjustments to the preload nut. You can't hurt the bearings on a Sherline spindle by letting them get too hot for a few moments. If it gets too hot to hold your hand on the headstock comfortably, back the preload nut off less than 2º and give the pulley end of the spindle a sharp tap with a mallet. One degree equals 0.00012". The tap is to move the inner race of the bearing on the spindle shaft increasing endplay. If you have a good indicator you can check the endplay by putting around ten pounds of pressure to both ends of the spindle, one end at a time, and reading the total movement. You could also spin a free running spindle with a flick of the wrist and if it makes less than two revolutions with a 3-jaw chuck mounted on it, it will probably need backing off to run for a long period at 10,000 RPM.

Replacing the standard pulleys with the 10,000 RPM pulley set

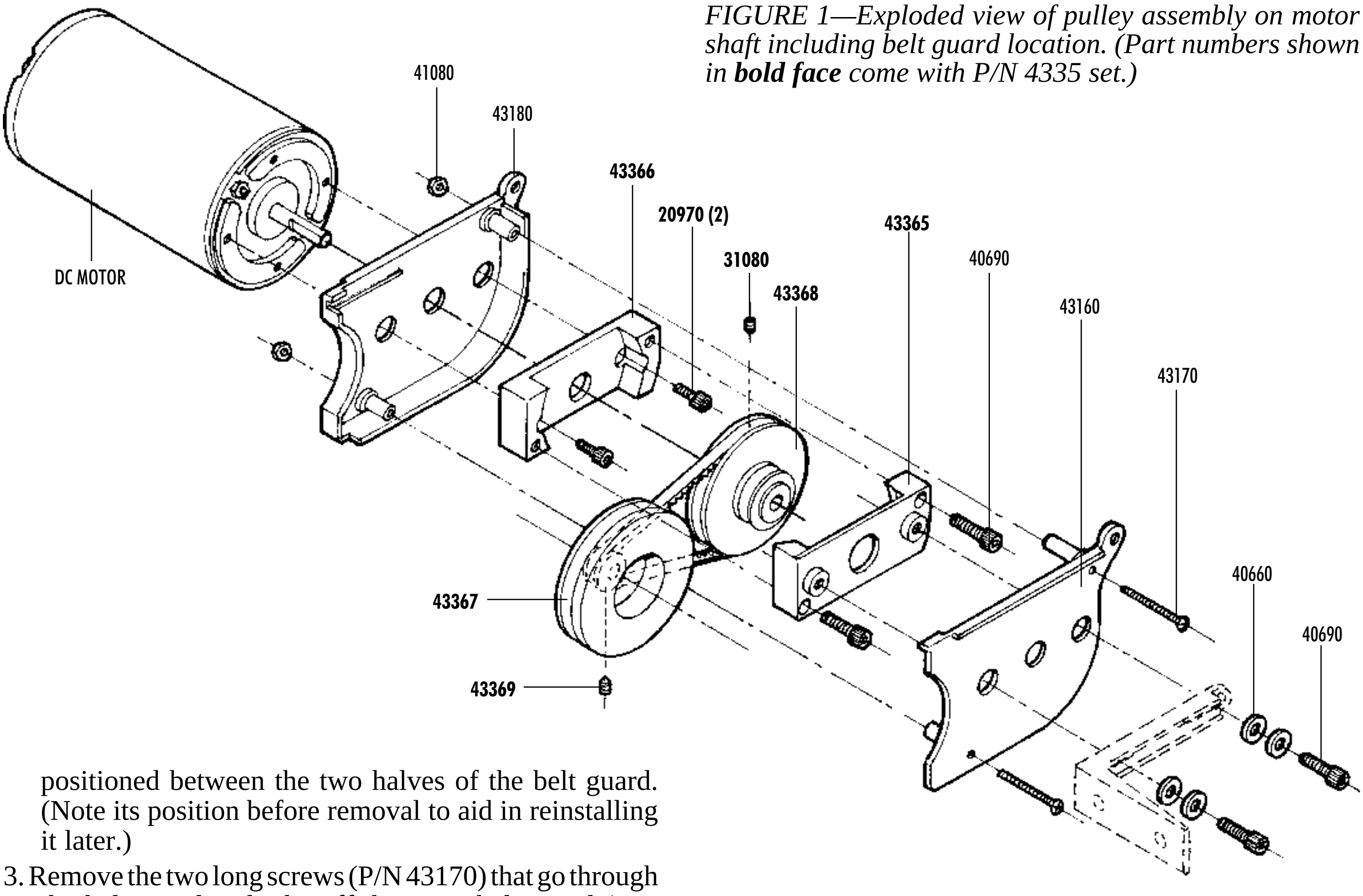
To remove the existing standard pulley set on your Sherline headstock and replace it with the 10,000 RPM pulley set, follow the instructions below. Refer to Figure 2 on page 4 of these instructions or to the exploded view in your *Sherline Assembly and Instruction Guide* for reference to the standard pulley and speed control parts.

Removing the standard pulley set:

Because the motor pulley diameter had to be larger than the center distance of the motor mounting holes, a relatively complex set of parts had to manufactured to replace the two inexpensive motor standoffs. These parts have to be assembled in the proper sequence in order to work properly.

1. Remove the two socket head screws and washers that hold the motor in position on the mounting bracket. Slip the drive belt off the spindle pulley, remove the motor and speed control unit and set it down on a folded towel or padded surface to work on it.
2. Remove the socket head screw that holds the speed control housing in place and pivot the speed control upward on its hinge pins to expose the motor pulley. Lift out the cover mounting plate (P/N 43130), which is

FIGURE 1—Exploded view of pulley assembly on motor shaft including belt guard location. (Part numbers shown in **bold face** come with P/N 4335 set.)



positioned between the two halves of the belt guard. (Note its position before removal to aid in reinstalling it later.)

3. Remove the two long screws (P/N 43170) that go through the belt guard and take off the outer belt guard (P/N 43160). (Make sure you don't lose the nuts [P/N 41080] that are inset into the rear belt guard. They may stick in the holes or they may fall out.)
4. Remove the drive belt. Using the smallest hex key provided with your machine, loosen the set screw in the motor pulley (P/N 43360) and remove it from the motor shaft.
5. Using an adjustable wrench, remove the two motor standoffs (P/N 43100). The inner belt guard (P/N 43180) can now be removed.
6. Loosen the set screw and remove the spindle pulley (P/N 43230) from the spindle shaft on the headstock.

Installing the new 10,000 RPM pulley set:

1. Slip the inner belt guard (P/N 43180) over the motor shaft and align the two outer holes with the appropriate holes in the end of the motor. See the exploded view for orientation of the motor and which two of the four holes to use.
2. Slip the inner standoff half (P/N 43366) over the motor shaft and attach it to the motor using the two 8-32 screws (P/N 20970). (Tighten using the 9/64" hex key provided with the kit.) Make sure the bosses on the back side register in the holes in the belt guard and are fully seated.
3. Put the drive belt over the motor pulley (P/N 43680) and slip the motor pulley and belt over the motor shaft. Tighten the set screw against the flat of the motor shaft. The pulley should be positioned close to the standoff but not touching it. Look at the exploded view in Figure 1 above for correct orientation of the pulley. NOTE: The pulley position can be readjusted and the setscrew can be firmly tightened in place after the motor is mounted to the headstock to assure proper belt alignment.
4. Attach the outer half of the standoff assembly to the inner half using the two 10-32 x 3/4" socket head screws (P/N 40690). Make sure the drive belt exits in the proper location, with the leg of the standoff going through the center of the belt circle.
5. Register the outer belt guard (P/N 43160) over the raised bosses on the outside of the standoff assembly and attach it to the inner belt guard using the long screws (P/N 43170) going into the inset nuts (P/N 43180) in the back of the inner belt guard.
6. Put the new spindle drive pulley (P/N 43367) onto the headstock spindle shaft and secure it by tightening the set screw located in the groove of the smaller pulley. See Figure 1 for orientation.
7. Lift the motor assembly into approximate position. Make sure the drive belt is correctly fitted over the larger of the motor pulley grooves and is not binding, and then slip the other end of the drive belt over the smaller of the two pulleys on the spindle pulley.
8. Register the holes in the end of the outer standoff housing with the slots in the motor mounting bracket and loosely install the two mounting screws with two washers on each 10-32 x 3/4" screw.
9. Push the motor assembly away from the spindle to put sufficient tension in the drive belt and tighten the screws in the motor mount to hold it in position. The belt does

not have to be extremely tight to work properly. Pressing on the belt with your finger halfway between the pulleys, you should be able to depress it about 1/4" when properly tightened. Turn the motor by hand to make sure the belt is not rubbing anywhere and whole assembly turns easily.

10. Insert one of the speed control housing hinge pins into one of the “ears” on the rear of the belt guard. Push to bend the ear slightly until the other pin can be seated in the other ear.
11. Install the mounting bracket (P/N 43130) in the slots in the two halves of the belt guard by setting it atop the two molded-in rails. It should slide back and forth slightly.
12. Pivot the speed control housing downward and attach it to the mounting bracket with the socket head screw and washer to secure it in place.

Changing speed ranges with the 10.000 RPM pulley set

If you need to change pulley speed ranges on the standard pulley set, it is done by simply pivoting the speed control out of the way, loosening the motor mount and slipping the drive belt from one pair of pulleys to the other. The 10,000 RPM pulley set, on the other hand, has very little clearance between the belt and the inside of the motor standoff pair. Therefore, it is necessary to remove the outer belt guard and outer standoff half and loosen the motor pulley to move the belt to the lower speed position. Here’s how:

1. Loosen the cap screws that hold the motor and speed control to the motor mounting bracket, remove the drive belt from the spindle pulley and set the motor unit down on a padded surface to work on it.

2. Remove the speed control mounting screw and pivot the speed control box upward.

3. Slide the mounting tab from between the belt guard halves, noting its position for later reinstallation.

4. Remove the long screws P/N 43170 that hold the two halves of the belt guard together and take off the outer belt guard.

5. Remove the two P/N 40690 socket screws from the outer standoff (P/N 43365) and take off the outer standoff.

6. Loosen the set screw (P/N 31080) in the groove of the motor pulley and slide it far enough down the motor shaft so that the pulley can be switched to the smaller pulley.

7. Move the pulley back into position and retighten the set screw.

8. With the drive belt in place on the smaller pulley, reinstall the outer standoff (P/N 43365) making sure the belt exits in the proper location.

9. Reinstall the outer belt guard (P/N 43160) using the two long screws.

10. Slip the drive belt over the large diameter groove of the spindle pulley and remount the motor/speed control unit to the machine. Push the motor outward on the motor mount to tension the belt as the two mounting screws are tightened.

11. Reinstall the speed control housing pins between the “ears” of the two halves of the belt guard.

12. Reinstall the mounting tab and swing the speed control housing down into place. Attach with the mouunting screw.

Pulley speed ranges

The standard pulley set offers a speed range of 70-2800 RPM in normal position and 45-1400 in the “hi torque” position. The 10,000 RPM pulley set actually offers a speed range of from about 1500 to 10,200 RPM in the high speed position and 150 to 2200 RPM in the low speed position. Maximum RPM in the high speed position can be affected by the preload setting. If the preload is set to the original factory setting of 0.0002" of runout, it may not be possible to achieve an actual 10,000 RPM. Loosening the preload to the recommended high speed setting of 0.0003" of runout should make speeds of 10,000 RPM possible.

—Joe Martin
President and owner

PRECAUTIONS

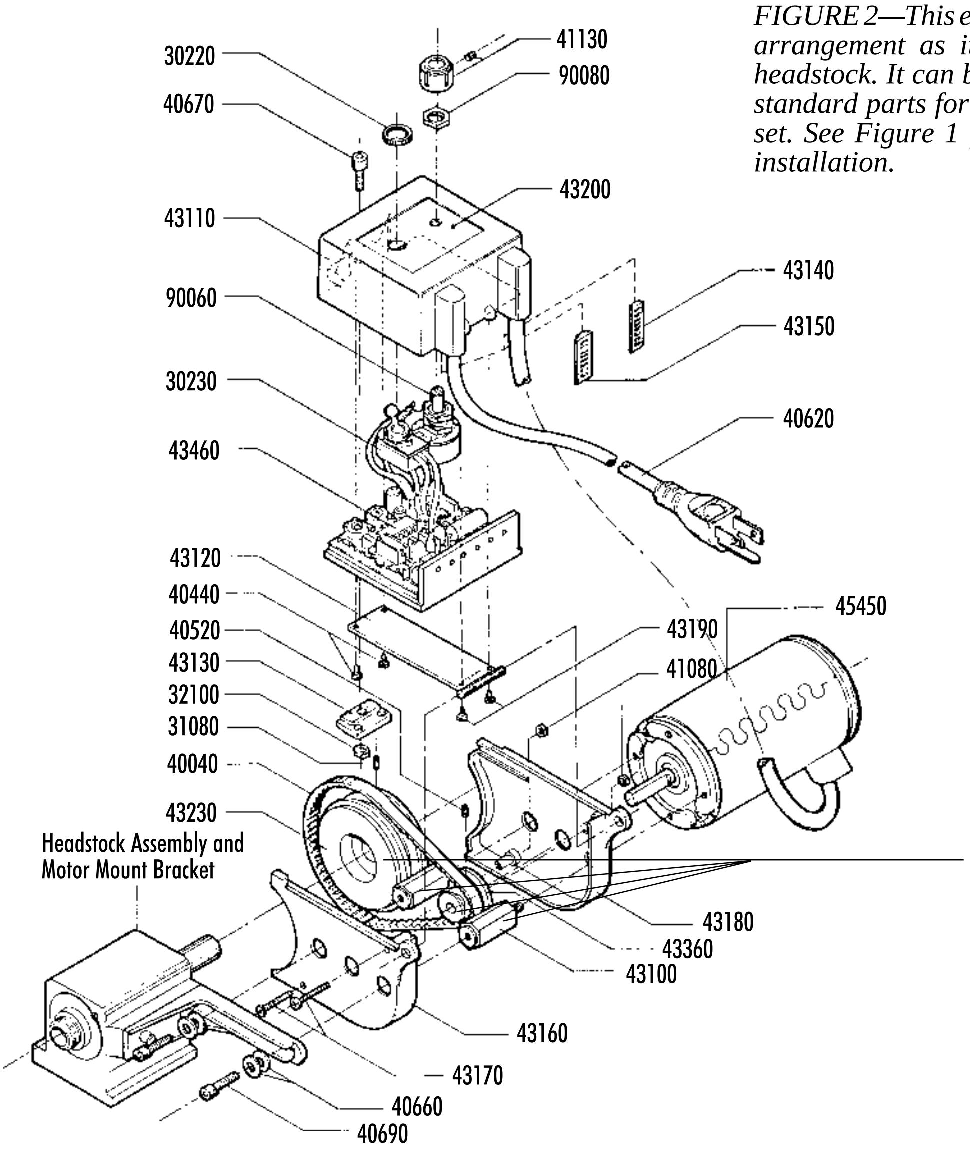
• Make sure speed control knob is in the slowest position before turning motor switch on, then adjust spindle speed as needed.

• Parts or chucks rotated at high speed must be in balance. Do not operate at high speed in an out-of-balance condition. Also, do not spin a chuck that is not tightened on a part as the scroll could unwind allowing the chuck jaws hit the lathe bed.

• Shafts at either end of the spindle should be supported if rotated at high speeds. An unsupported shaft that is slightly out of center can suddenly whip and bend 90° if speed is high enough.

• If headstock becomes too hot to hold your hand on it, the preload is set to tight. Refer to the instructions to back off the preload nut slightly.

FIGURE 2—This exploded view shows the standard pulley arrangement as it comes with the regular motor and headstock. It can be used to help you when removing the standard parts for installation of the 10,000 RPM pulley set. See Figure 1 for the parts that are different in that installation.



NOTE: These two standard pulleys and motor standoffs are replaced by the new pulleys and standoffs when converting a standard headstock/ motor/speed control unit to the 10,000 RPM pulley set

PARTS LISTING

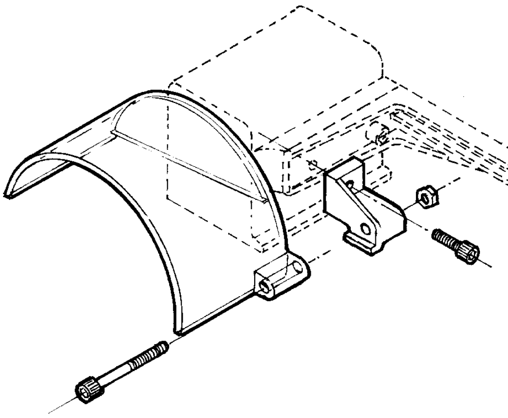
Parts for P/N 4335 10,000 RPM pulley set

P/N	REQ.	DESCRIPTION
12070	1	9/64" Hex key (not shown)
20970	2	8-32 x 1/2" SHCS
31080	1	10-32 x 3/8" Flat point set screw
40690	2	10-32 x 3/4" SHCS
43365	1	Outer standoff half
43366	1	Inner standoff half
43367	1	10,000 RPM Spindle pulley
43368	1	10,000 RPM Motor pulley
43369	1	10-32 x 1/4″ cup point set screw

For P/N 33070 with headstock, motor and control add the following parts

P/N	REQ.	DESCRIPTION
45450	1	DC motor w/ external brushes
41080	2	6-32 nut
43180	1	Inner belt guard
43130	1	Speed control cover mounting plate
32100	1	10-32 hex nut
43120	1	Speed contol hinge plate
43110	2	Motor standoff
43160	1	Outer belt guard
43170	2	6-32 x 1-3/8″ pan head screw
43140	1	Speed control tab, small
43150	1	Speed control tab, large
43460	1	Speed control electronics
43110	1	Speed control case

P/N	REQ.	DESCRIPTION
90060	1	Speed control potentiometer
90080	1	Potentiometer washer
41130	1	Speed control knob with set screw
30230	1	On/off switch
30220	1	Switch knurled round nut
40670	1	10-32 x 1/2″ SHCS
40660	5	3/16″ I.D. washer
43200	1	Speed control label
40620	1	8′ power cord, USA (or 80630, UK; 40640, Europe)
40040	1	Drive belt
40520	1	Preload nut
40420	2	Headstock bearing
40100	1	Headstock case
40230	1	Spindle
40320	1	Bearing washer
40020	1	Motor mount
40510	2	10-32 x 3/8″ SHCS
40690	2	10-32 x 3/4″ SHCS
40520	1	10-32 x 3/16″ cup point set screw
40540	1	5/16-18 x 3/4″ cone point set screw
43190	2	#2 x 14″ flat head sheet metal screw
40440	2	Self tapping screw



WEAR YOUR SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING

UNITurn & UNIMill
The cool tool

CHIP GUARD
P/N 4360

WARNING! THE CHIP GUARD IS NOT INTENDED TO ELIMINATE THE NEED FOR SAFETY GLASSES.

Always wear safety glasses whenever operating machine tools.

The Chip Guard was developed both as a safety feature and to help keep your work area cleaner by containing flying chips to a smaller area. (This will be a particularly welcome feature for those who operate their UNITurn lathe on the kitchen table or anywhere inside the house.) It mounts easily and swings up out of the way for easy access to the Headstock when setting up a job. It is molded from clear polycarbonate material to make it as easy as possible to see what you are doing yet still be protected while working. This material is used because it is strong and resistant to impact. However, solvents are very hard on it, so only use mild soapy water to clean it. Material selection is always a compromise, and we believe this to be the safest choice.

MOUNTING THE CHIP GUARD

From the Motor Bracket, remove the 3/8" Socket Head Cap Screw closest to the Spindle end of the Headstock. Place the Chip Guard Hinge between the flanges of the Bracket and run the longer 5/8" Socket Head Cap Screw provided through the hole in the Chip Guard Hinge and back into the original hole in the Headstock. The clear Chip Guard will now rotate on the pivot screw to move up and out of the way when setting up a job or back into place before turning on the motor.

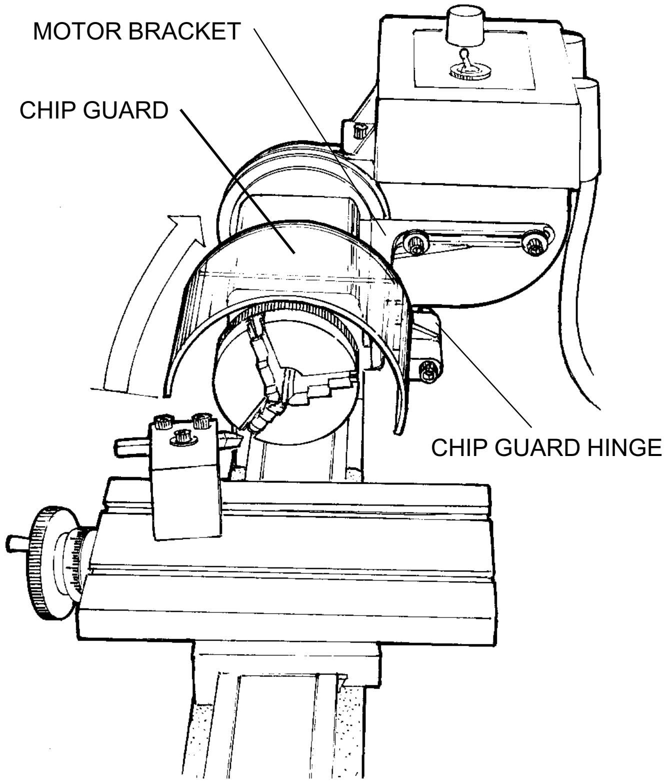


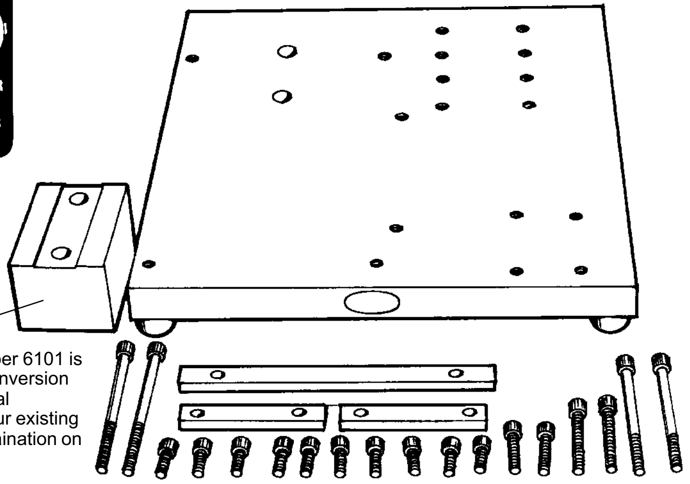
FIGURE 1—The Chip Guard mounted on a UNITurn Lathe. The guard pivots up out of the way for access to the spindle area.

REPLACEMENT PARTS LIST

NO. REQ.	PART NO.	DESCRIPTION
1	4361	Chip Guard
1	4362	Chip Guard Hinge
1	4033	10-32 x 5/8" Skt. Hd. Cap Screw
1	4070	10-32 x 1-3/4" Skt. Hd. Cap Screw
1	3210	10-32 Nut

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

UNITurn & UNIMill
The cool tool



NOTE: Part Number 6101 is not included in Conversion set. It is an optional modification of your existing column. See explaination on page 2.

HORIZONTAL MILLING CONVERSION
P/N 6100

With the **HORIZONTAL MILLING CONVERSION** (P/N 6100), the mill spindle (in the horizontal position) can be aligned with the X and Y axis. There are three places the column can be mounted to the HORIZONTAL CONVERSION base. When the spindle is lined up with the “Y” axis the outer most position is for drilling and milling (see POSITION A).

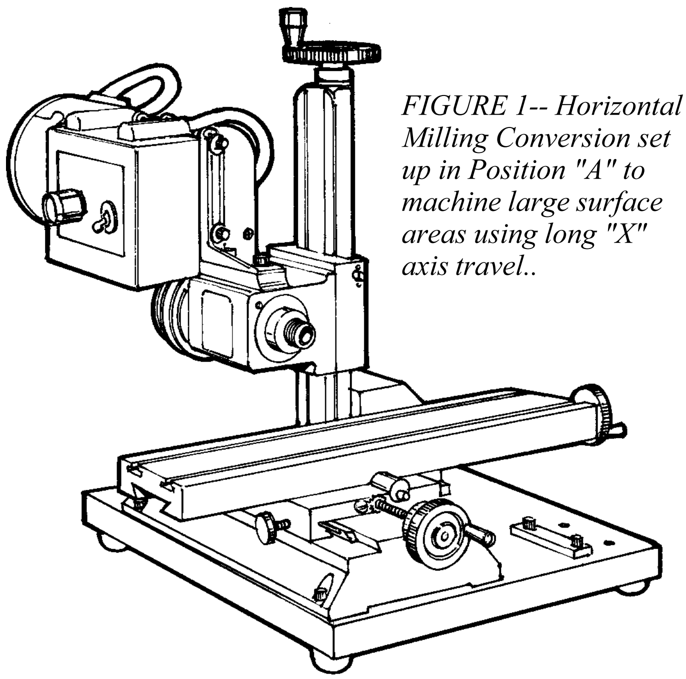


FIGURE 1-- Horizontal Milling Conversion set up in Position "A" to machine large surface areas using long "X" axis travel..

The closest position is used for milling. The configuration of the work has a lot to do with the choice to be made, but remember when milling, the closer the end mills are mounted to the spindle bearings, the more rigid the set up. The spindle can also be mounted lined up with the “X” axis by reversing the “XY” table on the horizontal base and mounting the column in the single set of holes.

To configure the machine so the spindle is over the “X” axis, the “XY” base must be reversed from its normal position, that is, with the “Y” axis handwheel away from the UNITurn label (see POSITION B).

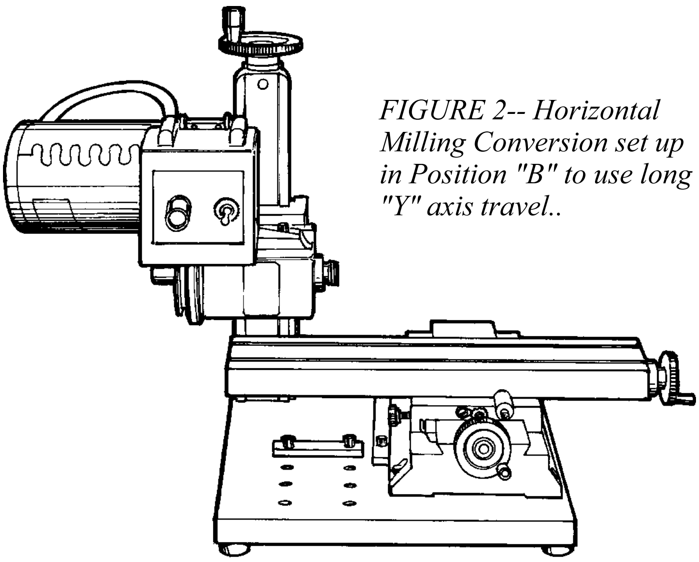


FIGURE 2-- Horizontal Milling Conversion set up in Position "B" to use long "Y" axis travel..

The advantage of this set up is you have 9" of throw from the spindle nose and you could drill and bore a hole 8" from the clamped down edge. If the mill was in its vertical configuration the same edge would interfere with the column. A point to consider is that any axis that moves the work in and out from the end of the spindle becomes the “Z” axis and the up and down of the column will usually be called the “Y” axis when the mill is in a horizontal configuration.

The 1/4" x 1/2" alignment bars are clamped against the column base and “XY” table after the machine is aligned so it isn’t necessary to align it every time the configuration is changed. How close the machine has to be aligned is dependent on the work to be performed. A machinist square from the milling table to the column bed (dovetail) will usually be good enough, but a dial indicator would be helpful for close tolerance work.

It is possible to move the “Y” axis saddle to the point the

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

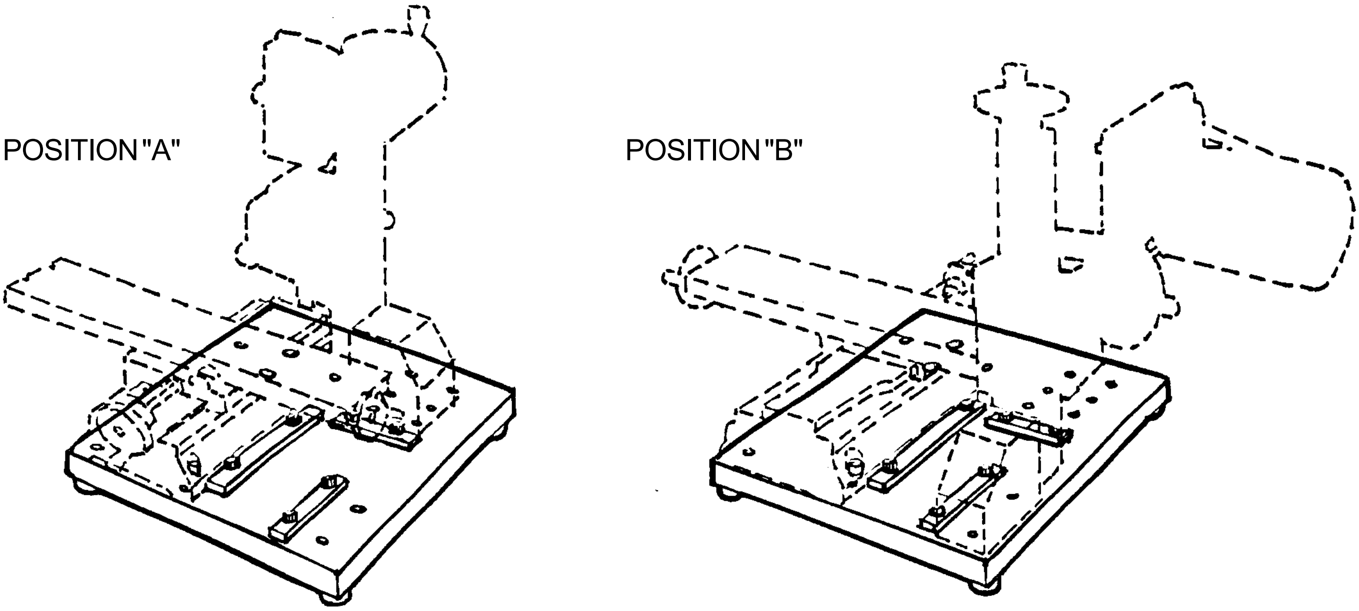


FIGURE 3-- Shows suggested position of Mill and alignment bars mounted to Milling Conversion Table for each position. Note that in Position "A", the vertical column and drive can also be mounted further toward the back of the table should your set-up require it.

lead screw will disengage from the nut without the column being in place (normally, it would hit the column base before it could disengage).

The advantage of modifying the column (P/N 6101) for this attachment is to allow the spindle center to go below the top of the table. This allows a piece of material to be clamped directly to the table and machine the edge overhanging the table. We also modify the column saddle with another alignment groove in the horizontal position. All vertical milling machines manufactured after 1991 will come with the groove cut. The column base is modified by cutting 2" off and making it a spacer block and retapping what's left over. This allows the column to be mounted with or without the spacer in either the horizontal or vertical configuration. If you have access to a saw and mill, you could make these modifications yourself. The drawings are included for these modifications.

A part held vertical with the right angle plate can have a 9" x 6" area that can be machined without moving the work. If you think about it, that's a lot of movement for a machine of this size.

We believe you will find this a useful accessory. The RIGHT ANGLE PLATE (P/N 3701) will also be very useful with the HORIZONTAL MILLING CONVERSION.

CLAMPING INSTRUCTIONS

To clamp the column to the HORIZONTAL MILLING CONVERSION plate, use the 1" x 1/4-20 Socket Head Cap Screw (SHCS) - (2 included)—use the 3" x 1/4-20 SHCS (2 included) when the spacer block is used.

To clamp the column to the "XY" base, use the 1-3/4" x 1/4 - 20 SHCS (2 included). Use the 3-3/4" x 1/4-20 SHCS (2 included) when the spacer block is used.

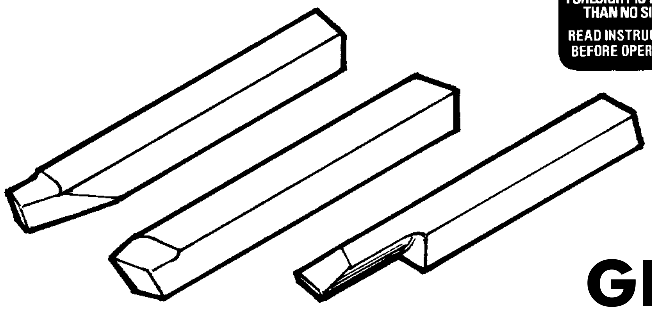
The alignment bars and the "XY" table are held to the base with 5/8" x 1/4-20 SHCS (10 included).

HORIZONTAL MILLING CONVERSION (P/N 6100) REPLACEMENT PARTS LIST

NO. REQ	PART NO.	DESCRIPTION
1	4056	3/16" Hex key
2	5022	1-3/4" X 1/4-20 Skt Hd Cap Screw
1	6102	Horizontal Mill Base
1	6103	6.3" Alignment Bar
2	6104	2.8" Alignment Bars
4	6110	Rubber Feet with 1/2" x10-32 Skt Hd Cap Screws
10	6111	5/8" x 1/4-20 Skt Hd Cap Screws
2	6112	1" x 1/4-20 Skt Hd Cap Screws
2	6113	3" x 1/4-20 Skt Hd Cap Screws
2	6114	3-3/4" x 1/4-20 Skt Hd Cap Screws
1	6115	Horizontal Milling Instructions

UNITurn & UNIMill

The cool tool



Right Hand Cutting Tool   Left Hand Cutting Tool   Boring Tool

GRINDING YOUR OWN LATHE TOOLS

As with any machining operation, grinding requires the utmost attention to “Eye Protection”. Be sure to use it when attempting the following instructions.

My first experience in metal cutting was in high school. The teacher gave us a 1/4" square tool blank and then showed us how to make a right hand cutting tool bit out of it in a couple of minutes. I watched closely, made mine in ten minutes or so, and went on to learn enough in one year to always make what I needed. I wasn’t the best in the class, just a little above average, but it seemed the below average students were still grinding on a tool bit three months into the course. I believe these students didn’t have the confidence in themselves to work with their hands. Grinding lathe tools is easy, and the only reason we sell them is to help a beginner get started. If you are to be successful in making metal parts on a lathe, you have to teach yourself to grind tool bits.

Consider a Carpenter who didn’t have the confidence to drive a nail because he was worried about missing the nail and hitting his thumb. He/she wouldn’t be in the trade very long! Some things you do in trades require a positive approach and tool grinding is one of them. If you keep stopping to see if you’re grinding it correctly you’ll not only waste a lot of time, but will end up with a less than perfect cutting edge. Set up the grinder correctly and do it! It shouldn’t take but a few minutes to make simple cutting tools and only a few seconds to resharpen them.

A bench grinder doesn’t have to be expensive to work well, but it does require good “wheels” for high speed steels. Try to find a source for grinding wheels from an industrial supplier. Some of the wheels that come with inexpensive grinders wouldn’t sharpen a butter knife. Sixty grit is a good place to start. A wheel dresser is also a necessity. They cost less and are readily available from good hardware stores.

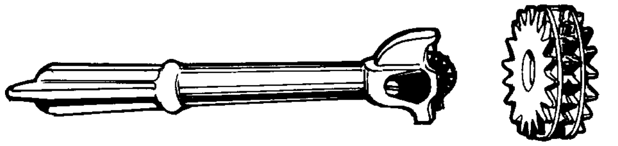


FIGURE 1-- A Wheel Dressing Tool and spare "star wheel " sharpening insert.

Grinding wheels should be considered cutting tools and have to be sharpened. A wheel dresser sharpens by “breaking off” the outer layer of abrasive grit from the wheel with star shaped rotating cutters which also have to be replaced from time to time. This leaves the cutting edges of the grit sharp and clean.

A sharp wheel will cut quickly with a “hissing” sound and with very little heat by comparison to a dull wheel. A dull wheel produces a “rapping” sound created by a "loaded up" area on the cutting surface. In a way, you can compare what happens to grinding wheels to a piece of sandpaper that is being used to sand a painted surface; the paper loads up, stops cutting, and has to be replaced.

For safety, a bench grinder should be mounted to something heavy enough so it will not move while being used. The tool support must be used and should be set at approximately 7°. Few people have the skill to make tools without a tool support and in essence it’s wasted effort. Tool supports are usually made up of two pieces that allow you to set your tool rest above or below center. It really doesn’t matter whether its above or below as long as the support is at 7°.

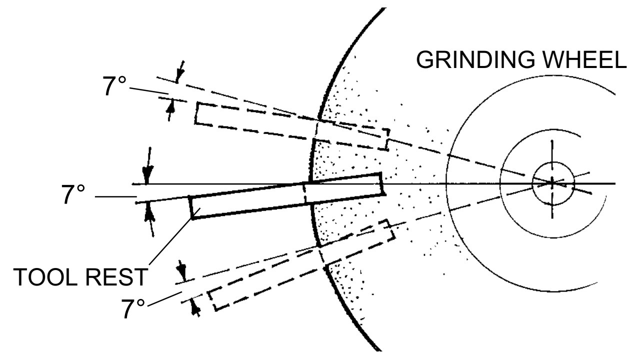


FIGURE 2--Set tool rest at any height, but at 7° angle from centerline of wheel.

The reason tool supports are designed like this is so they can be used for a variety of uses, not just tool bits. What this means is that if the tool support is above or below center it must be adjusted as the wheel diameter changes.

The cool tool®
www.thecooltool.com
Modellbauwerkzeug & Präzisionsmaschinen G .m.b.H.
Modelmaking & Precision Tools Ltd. Vienna / Austria
Fabriksgasse 15,A-2340 Mödling info@thecooltool.com
phone:+43-2236-892 666 fax: +43-2236-892666-18

Now it's time to make a tool, and whether you turn this job into a major project is up to you!

When working around grinders it is an absolute necessity to wear EYE PROTECTION. Grinding debris is thrown out at high velocities and can damage not only eyes, but also expensive glasses. Wear safety glasses or a full face shield.

If you've never sharpened a tool, take a close look at how ours are sharpened. Let's duplicate the right hand tool on the opposite end of the blank. Be careful you don't cut yourself on the blank or the sharpened end while working with it.

First dress the wheel by taking the dresser and setting it on the tool support square with the wheel and while applying a light pressure move the dresser back and forth with the grinder running. Unless the wheel is in bad shape, it should be ready to use in a few passes.

GRINDING SIDE 1 OF THE TOOL

Turn off the grinder and set the tool support for approximately 7° if you haven't done it yet. If you're not good at guessing at angles use a presharpened UNITurn tool to set the angle. Metal cutting tools are very tolerant on angles. I've always found wood cutting tools more difficult to sharpen. Too little angle and the "heel" of the tool will rub, too much angle will cause the tool to "dig in" and chatter.

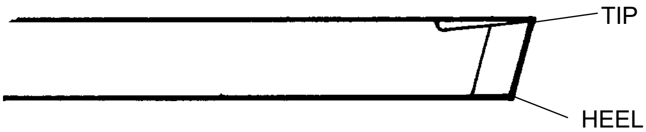


FIGURE 3-- Heel of tool.

Have a cup of water handy to cool the tool with and set the blank on the tool rest and start grinding side 1.

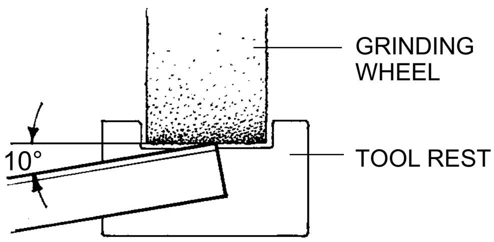


FIGURE 4--Grinding Side 1.

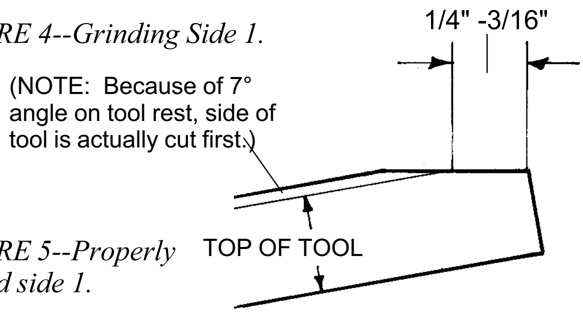


FIGURE 5--Properly ground side 1.

Move the blank back and forth across the face of the wheel until you have ground a 10° angle on approximately 3/16" (4mm) of side 1.

This is where the "positive approach" comes in. Unless you push the tool into the wheel with enough pressure, the tool will bounce around and you'll never get a good flat cutting surface. It isn't necessary to worry about getting the tool too hot. Modern day tool steels don't anneal and a little discoloration doesn't effect the tool life in tool room use. What you should worry about is not burning yourself or grinding the tips of your fingers off! Concentrate on holding the 10° angle while moving back and forth. We'll give this edge a final sharpening later; it's time for side 2.

GRINDING SIDE 2 OF THE TOOL

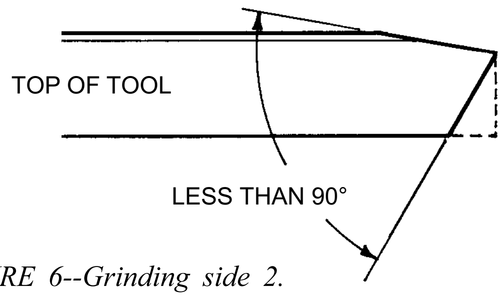


FIGURE 6--Grinding side 2.

The reason angle B is ground less than 90° is to allow the tool to get into corners.

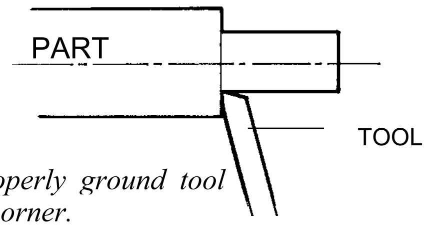
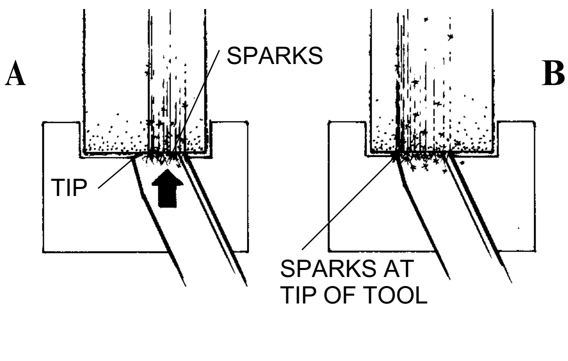


FIGURE 7--Properly ground tool cutting into a corner.

Side 2 is ground the same way as side 1, moving the tool back and forth until you have a point. After you get side 2 ground, cool the tool in the cup of water.

Now I want you to learn another aspect of tool grinding. It's important to know when you have ground the surface up to the cutting edge, especially when resharpening lathe tools. Take the tool you just ground and bring it up to the wheel at a slightly different angle than you just ground for this experiment. Watch the point that touches the wheel first



FIGURES 8A-- Tip not yet ground flat and 8B, Tool ground flat all the way to the tip.

and you will notice that the sparks will bounce off the cutting edge only where the wheel has ground from top to bottom.

This tells you when the tool has been sharpened without taking it away to look which allows you to grind flat and true surfaces. If you sharpen a tool for a SHERLINE lathe, use a 1/4" square tool blank and keep the cutting edge up to the top of the blank; the tool will come out on center without shims. You will have to be precise grinding the third side to accomplish this.

GRINDING SIDE 3

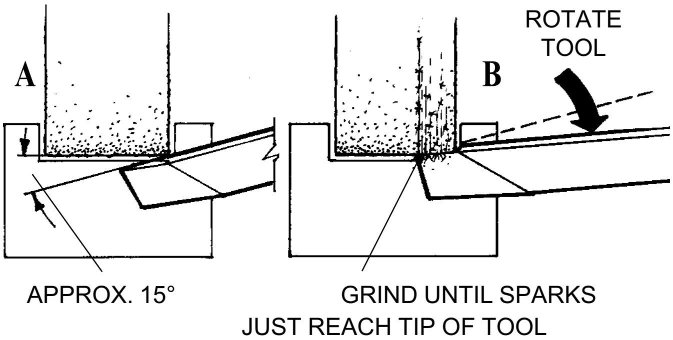
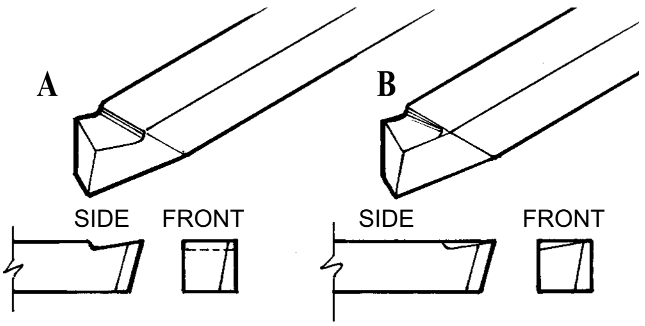


FIGURE 9--Grinding the "Hook" into side 3.

Use the skill you have developed grinding the second side now. Set the blank on the support with the 10° (side 1) up. The tool has to be brought up to the grinding wheel with a slight angle so you don't grind the tip below center. With the tool setting on the rest, move the tool in and grind until you see sparks bouncing off the cutting edge where the corner of the wheel is lined up with the back part of the 10° face. When this happens, slowly decrease the angle without pushing the tool in any more until sparks bounce all the way to the tip. Stop as soon as this happens. You may inspect it, and the surface should be entirely ground. The recommended way is to put more "hook" on the tool than I have suggested, but I have found that the slight increase in performance is offset by the problems encountered resharpening these tools.



FIGURES 10A--Normally recommended "hook" ground into tool and 10B, Simpler method suggested for Uniturn tools.

To put the finishing touches on your tool, you have to "kiss off" sides 1 and 2 again. You must carefully line up side 1 with the wheel and bring it to the wheel in a positive manner with very little pressure; watch for the sparks on the cutting edge. What you're trying to accomplish is to make the tool set against the wheel on the same plane as when you first ground side 1. If the tool is held too rigid, it will not align itself, too loose and it will bounce around.

"BREAKING" THE POINT

Use the same method on side 2. The tool should be ready to use except for the point. I always put about a .010 (.2mm) "break" on the point by holding the tool with the point aimed at the wheel face. Because two angles converge at the point, the angle in relation to the sides is greater. Think about it!

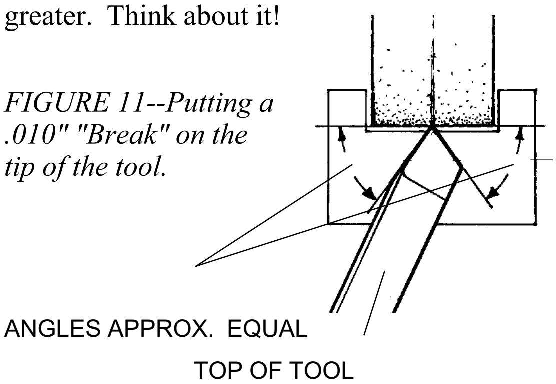


FIGURE 11--Putting a .010" "Break" on the tip of the tool.

This means that if you set the tool flat on the tool rest the tool rest angle would have to be increased to get an even flat. This wouldn't be worth the effort, so the easy way is to free hand it. I always start by touching the heel of the tool first, and then change the angle until a slight flat is put on the tip. Of course, the angle you're holding it at has to be close when starting to get desired results.

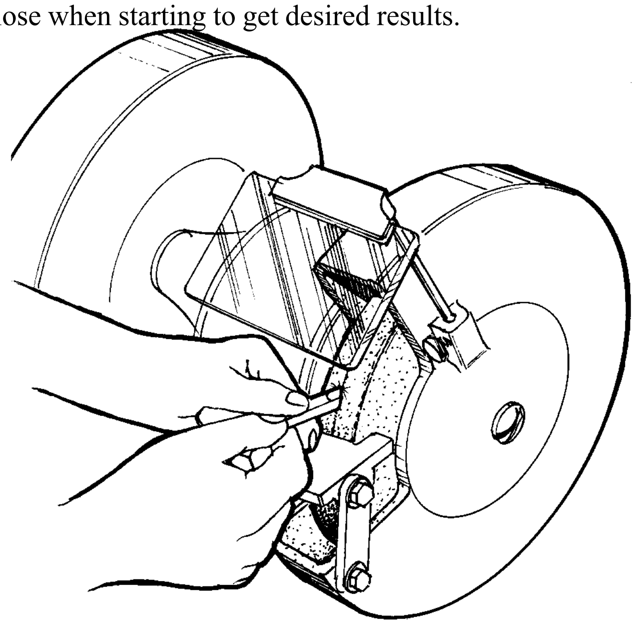


FIGURE 12--Handholding the tool to "Break" the point saves resetting the angle on the tool rest.

The purpose of this flat is to improve finish and tool life. I don't recommend a large radius on the tip of tools used on small machines. These machines are not rigid enough to get the desired results from this practice and cause "chatter" problems.

The finished product should be a right handed tool, have flat cutting surfaces (except for the radius caused by the wheel), have a slight flat on the tip, and a tip angle of less than 90°.

Tools used on lathes such as the UNITurn will do all their cutting at the tip of the tool because they don't have the horsepower for 1/4" (6 mm) cuts.

I don't recommend using oil stones to improve the edges. After a few minutes use with an occasional dab of cutting oil a properly sharpened tool will hone itself in.

I always believe the final sharpening to a tool should take place with the wheel cutting the cutting edge of the tool from the top of the tool to the bottom when using bench grinders.

I realize I've given a great deal of information on how to do what I call a simple operation, but these are very complex instructions to write because I'm trying to tell you how to control your hands, not a simple machine.

Incidentally, the reason we call a tool a right handed tool when the cutting edge is on the left is because it is designated by which way the chip leaves the cutting tool. Cutting tools such as left or right handed tin snips are also designated in this manner because the cut-off falls to the left or right.

The left hand tools are ground the same as right, in the same order with the angles reversed.

BORING TOOLS

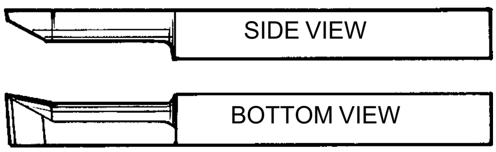
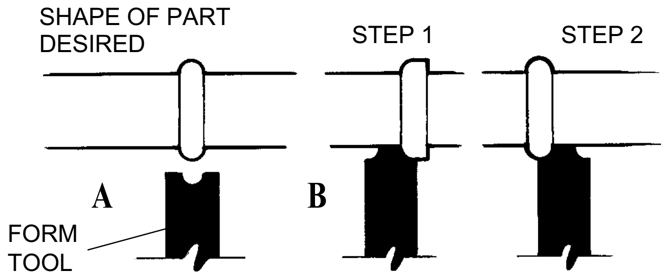


FIGURE 13--Typical Boring Tool.

Boring tools are the most difficult to grind. They should always be made as rigid as possible. Tool angles around the "tip" can be the same as any cutting tool, but clearances of the tool body have to be considered carefully. A tool ground with enough clearance for a finished hole may not have enough clearance to start with when the hole has a smaller diameter. If you have to bore a hole in a part that has a lot of work in it, have a tool ready to use that's been checked out on a piece of scrap.

FORM TOOLS

Form tools are used to create a shape the same as the tool. To grind form tools, a pattern of the finished shape should be at hand and there should be some possibility of success with what you have to work with. You can't grind a 1/8" (3 mm) groove into your tool 1/4" (6 mm) deep with a 1/2" (12 mm) wide wheel.



FIGURES 14A--A Typical Form Tool made by a custom toolmaking shop and 14B, a home shop method of achieving the same finished shape in two steps with a tool that can be ground on a bench grinder.

This type of tool is usually made by Tool and Cutter specialists that have high shop rates using precision grinders, diamond dressers, and a large variety of wheels available to them.

All is not lost if we have a good pair of hands with a good mind driving them! We can use the grinding wheel corners on our grinder and generate the shape 1/2 at a time on each side of the tool and still get our job done.

Form tools don't need any top relief (hook) to work. Use low spindle RPM and steady feed rates to prevent chatter. The width of a form tool should never exceed three times the smallest diameter of the finished part.

Like any skill, tool grinding is one that has to develop with time. It is also the skill that allows you to go one step beyond the average hacker.

H.S. TOOL BITS AVAILABLE FROM UNITurn

PART NO.	DESCRIPTION
1195	H.S. STEEL CUTTING TOOL, RIGHT
1196	H.S. STEEL CUTTING TOOL, LEFT
1197	H.S. STEEL CUTTING TOOL, BORING
1200*	H.S. STEEL INTERNAL THREADING TOOL
3005	H.S. STEEL 1/4" SQUARE TOOL BLANK
3005B	H.S. STEEL TOOL BLANKS (5-BULK)
3007	H.S. STEEL SET (RIGHT, LEFT, BORING)

*NOTE, the Internal threading tool is very difficult to make on just a bench grinder. If a precision thread is required, I recommend you buy our P/N 1200 which is preground.