CHOOSING & USING A MILLING MACHINE

Bench mill vs. Knee mill?

There are similarities between ALL small vertical milling machines and the traditional drill press, right:

1. A vertically-movable quill which encloses the spindle.
2. A drill press lever which propels the quill downward.
3. A quill clamp to lock the quill firmly in position.
4. A variable-speed spindle drive system.
5. A headstock that can be moved up or down on a vertical column.

Feature (5) applies only to bench mills, but check for this important difference: Many, but not all, bench mills have dovetails for headstock height adjustment, not round columns — see next page.

Every vertical mill is a part-time drill press, but there’s more to it than that. Comparing mills with drill presses, here are the KEY DIFFERENCES:

1. Massive, rigid construction, a lot more cast iron.
2. Heavy T-slotted movable table on dovetail ways, with ±0.0005" position measurement capability (optional 2- or 3-axis DRO).
3. The workpiece doesn’t slide on the mill table: instead it is firmly clamped to the table, which moves left-to-right and front-to-back in precise increments.

4. A mill spindle is designed for both down load (axial, like a drill press), and also side load (radial). That is why a mill spindle runs in tapered roller bearings (or deep-groove ball bearings) inside the quill.
5. The spindle isn’t just for drill chucks — use any R8 compatible device — end mill holders, collets, slitting saws, etc.
6. The headstock can be swiveled from left to right and (on knee mills) front to back.
Bench mill vs. Knee mill? (continued)

BENCH MILL
The bench mill (sometimes called a mill-drill) is what you might expect – a machine that can be bolted onto any rigid work surface, ideally 30 inches or more above the floor. Stands, some welded steel, some cast iron, are available for all bench mills offered by Precision Matthews (PM). Bench mills are similar in concept to the standard drill press, but with a host of additional features. The main differences between the two are the mill’s heavier, more robust construction, and how the workpiece is handled: On a drill press the workpiece is usually moved into position by sliding it on the table; On a mill it is rigidly clamped to the table, then positioned precisely by leadscrews.

Most bench mills have these basic features:
1. A saddle on dovetailed slides that can be moved in precise increments forward and backward by the Y-axis leadscrew.
2. A flat table on a second set of dovetailed slides, on top of the saddle, that can be moved in precise increments left to right by the X-axis leadscrew (X and Y axis dovetails in the saddle are at right angles).
3. Three or more T-slots on the table for clamping the workpiece or vise.
4. A column at the back of the machine, right angled to the table in both X and Y axes, that supports the headstock and allows its vertical position to be adjusted, usually by a handwheel and leadscrew. This is the Z-axis (on some bench mills the Z-axis is powered).
5. A motor-driven spindle running in a sleeve (the quill) that slides up and down about 4 inches within the headstock casting. Quill position is independent of the Z-axis headstock setting (unless the quill is locked to the headstock). The quill is controlled by a pinion that engages a rack of teeth on the quill; the pinion is controlled by an external handle, exactly as in a drill press. All PM bench mills have in addition a fine-feed handwheel to adjust quill position in very
Choice of spindle drive system

Just as in a drill press, the mill headstock supports the motor and the spindle drive system. The drive system in all small mills delivers spindle speeds from about 100 to 2000 rpm. Usable torque over such a wide range is not possible without some form of intermediate range selection, either through gears or two-step pulleys:

- **Gear head**, usually a six-speed gearbox (high and low ranges, three speeds in each range)
- **Belt drive**, usually available only for variable speed motors (high/low range selection is done by shifting the belt between pulleys)

A flared chamber at the bottom end of the spindle for R8 tool holders, examples in Figure X. The shanks of all R8 fittings are internally threaded to mate with the drawbar, next item.

7. A drawbar within the spindle to secure the R8 tool holder. The lower end of the drawbar is threaded UNF 7/16”-20.

Other features

Unlike the typical drill press, a bench mill headstock can usually be rotated and set at ± 90 degrees side to side relative to the zero-degree “drilling machine” axis used everyday, Figure 3. However, on a bench mill the headstock **cannot be rotated forward and backward** (however, this does not apply to most knee mills).

Leadscrew backlash is the main issue with dead-reckoning — a full turn of the handwheel may move the table by 0.1”, but a full turn in reverse doesn’t put the table back where it was. On the other hand, a DRO reports exact positions, completely unaffected by backlash.

### Power feed

Power feed for the table (X-axis only) is optionally available on all PM bench mills, Figure 4.

Digital Readouts (DROs) are available for most PM bench mills in two-axis (XY) and three-axis (XYZ) versions, Figure 5. If a three-axis DRO is installed, the Z-axis displays headstock elevation; this can be used as a measure of tool depth if (and only if) the quill is locked.

A DRO is not essential for precise machining, but it is truly a game changer. Once you have used a DRO, it is impossible to imagine how you managed without it. Without a DRO, the mill table is positioned by reference to the graduated dials on the handwheels, in other words by “dead-reckoning” (which is exactly how it was done, not so long ago, by every machinist, everywhere).

### Leadscrew backlash

Leadscrew backlash is the main issue with dead-reckoning — a full turn of the handwheel may move the table by 0.1", but a full turn in reverse doesn't put the table back where it was. On the other hand, a DRO reports exact positions, completely unaffected by backlash.
Bench Mill FAQ: Round or Square column?
Most drill presses — and some bench mills from other suppliers — have round columns, typically 3 or 4 inches in diameter. For basic drilling operations round columns are perfectly functional. However, for milling they can be troublesome, especially when tool swapping is necessary. One frequent instance of this (there are many) is reaming a just-drilled hole. The headstock must be raised, because the reamer is longer than the drill. How can that be done without affecting spindle alignment? With a round column, no way!

Today's solution to the alignment problem is the so-called “square” column, actually a dovetailed slide. This allows the headstock elevation to be changed by many inches with zero effect on the spindle axis relative to the workpiece. All bench mills from Precision Matthews have square columns.

KNEE MILL
The knee mill is similar to the bench mill in most respects except that its headstock cannot be vertically adjusted: in other words it has no column.

Z-axis motion comes instead from a movable knee, Figure 7, actually a very robust platform for the saddle and table components. Unlike the headstock on most bench mills, the knee can be raised in small, precise increments, allowing the cutting depth to be fine-tuned within ± 0.0005 inches, even less.

The knee mill headstock is usually mounted to the front of a ram, Figure 8, which allows it to be repositioned front to back. Also, in some cases, the ram sits on a turret that allows full-circle rotation in the vertical axis.

Just as on a bench mill, the knee mill headstock can be rotated and set at ± 90 degrees left or right relative to the zero-degree axis used for ordinary operations. On all knee mills — but not on bench mills — it can also be rotated forward and backward by a few degrees, certainly enough to allow precise 90 degree spindle-to-table alignment (known as “tramming”). In some cases the fore-aft rotatability is as much as ± 45 degrees.

Finally, perhaps the most obvious distinguishing feature of the knee mill is its weight. Even the smallest of knee mills from Precision Matthews is significantly heavier than PM bench mills, which start at about 300 lb). What's important about weight? In a word: rigidity. This is the key factor that determines maximum depth of cut and, to a lesser extent, smoothness of the cut surface. (That said, all bench mills can do a comparable job, simply by using less aggressive cuts and lower traverse speeds.)

Plus features of most knee mills:
• Rock solid, heavy, built-in stand
• Larger work envelope and greater flexibility than bench mills of similar table size
• Ram allows front/back positioning of the headstock
• Turret mounting allows ram rotation (not all cases)
• Rotatable headstock, ± 90 degrees left or right, up to ± 45 degrees fore and aft
• Power downfeed of the quill
• One-shot lubrication of machine ways
• Built-in spindle lock for easier tool changes
• Power feed usually available for all three axes: table (X, Y), and knee (Z)
WORKING WITH A NEW MILLING MACHINE

First, a possible mystery item ...
Import milling machines, especially bench mills, are sometimes shipped with an accessory not often seen in small shops. It is a shell mill arbor like this:

This example has an R8 shank and a 1" diameter arbor. It is intended for shell cutters like the one at right. It is perfectly functional, but may not be suitable for use with small machines lacking the rigidity and/or motor power for cutters 1-1/4 inches and more in diameter (another reason not to use it is the high cost of shell cutters).

Setting up
The following notes apply mostly to bench-style milling machines in small shops. It is assumed you have the mill cleaned, degreased, and set up on a solid foundation, either on its dedicated stand or on a bench with a solid top, 1-1/2" or more thick.

The stand or bench should rest on a substantial floor, ideally poured concrete. A lightweight mill on a dedicated stand is top heavy, so the stand should be bolted to the floor, using metal shims for leveling, see below. Heavier machines may be safely operated without bolting down, but there can be no guarantee — if in doubt, use bolts.

Using your most accurate level check the mill table in both axes, X left-to-right and Y front-to-back. If the mill is bolted to a dedicated stand, shim the base of the stand before bolting down on the concrete. If the mill is on a bench, shim under the base casting to avoid racking the machine when bolting down.

Why bother with leveling? The short answer is that machining accuracy doesn't depend critically on leveling, as it does with a lathe. That said, a level table can be a useful point of reference when setting up larger workpieces. Much more important than leveling, already mentioned, is not distorting the base casting by over tightening the hold-down bolts.

If you are using the mill for the first time ...
- Check the manual for power-up and tool installation procedures.
- As shipped, the headstock may be accurately enough aligned — vertical enough — for everyday drilling, but may require more precise setting up for milling.

ESSENTIAL ACCESSORIES (1) DRILL CHUCK
Vertical mills are always part-time drilling machines, so you will need a precision drill chuck. The one-piece style with integral R8 shank, Figure 10, is more rugged, with less run-out, than the more common variety with a separate arbor. (If you go with a two-piece chuck/shank, the chuck will likely come with a Jacobs taper cavity and the R8 arbor will have a matching male taper. Clean the tapers, then press the components together. With the chuck jaws retracted, secure the arbor by tapping it with a rawhide mallet.

To disassemble, use chuck removal wedges. These are a specialty item. You may get lucky using a pair of plastic construction-type wedges instead (cut U-shape slots in the ends to insert between chuck and arbor, Figure 9).
ure 12 (there are two sizes, 25 and 28 mm, about 1" and 2.2" diameter).

Figure 15 Clamping kit demo

To remove the chuck, unscrew the drawbar a couple of turns, then rap the top of the drawbar with a non-marring hammer, such as Figure 13.

Figure 13 Free the drawbar with a hammer like this

Fully unscrew the drawbar with one hand while supporting the R-8 device with the other hand. This is to prevent the chuck from falling out unexpectedly.

With the chuck installed the mill can be used like a drill press — simply slide the workpiece around on the table in the usual way. However, that's not a good idea unless you have protected the table with a thick piece of scrap material, e.g. MDF, and with the quill stop set to keep the drill well clear of the table (a new mill table peppered with drill holes is the saddest sight).

ESSENTIAL ACCESSORIES (3) CLAMPING KIT

In most machining operations the workpiece is firmly clamped to the table, which is moved relative to the cutting tool by the X and Y leadscrews. If the workpiece has at least two parallel sides a vise is the handiest means of clamping. Irregular shaped workpieces take more ingenuity, using various combinations of clamping kit items. Clamping kits are not one-size-fits-all, so be careful when purchasing — for instance, 1/2" diameter studs are too big for 12 mm T-slots (go with 7/16" or 3/8" instead).

Figure 14 Collet examples

ESSENTIAL ACCESSORIES (2) COLLETS

R8 collets, Figure 14, are hardened precision sleeves bored for various diameters from 1/16" to 7/8". They are installed and removed just like the drill chuck described above. The upper end of the collet is threaded 7/16"-20 for the drawbar, Figure 11. Slits in the collet allow a small degree of closure as the drawbar pulls the collet into the spindle taper, thus gripping the inserted cutting tool. A set screw on the inner surface of the spindle locates in a keyway on the collet to prevent rotation of the collet in the process of tightening. In cases where the grip is insufficient, a dedicated end mill holder may have to be used instead of a collet.

Take care when loosening the drawbar with the intention of removing a cutting tool. The tool may drop out unexpectedly, causing a lot of damage.
ESSENTIAL ACCESSORIES (5) VISE

For routine milling operations the workpiece is held in a precision vise. For small mills 4” is the most suitable vise size. The vise is secured to the table by T-bolts, Figure 19. When buying a vise, check the T-bolts that come with it will fit your mill’s T-slots.

Precision milling vises are distinguished from garden-variety machine vises by two main features:

The first of these is a special pusher device, propelled by the clamp screw, that exerts both forward and downward forces on the movable jaw. This is important because it eliminates the tendency of the movable jaw to lift as it tightens on the workpiece.

ESSENTIAL ACCESSORIES (4) DIAL INDICATOR

A dial indicator makes short work of many routine setup operations on the lathe, the mill and the surface plate. As with most model shop operations there are always alternative ways to get results, but a dial indicator is overall a good investment. It doesn't have to be a name brand, like the ones shown here, because in most cases we are comparing one setting with another, not making precise absolute measurements.

![Dial Indicator Schematic](image)

**Figure 16 Example dial indicators**

(1) Is the most popular style of dial indicator; imports from $20 and up. Most applications of style (1) call for dedicated holders, such as (2), a shop-made holder for use in a 1/4” collet. (3) Starrett’s Last Word indicator, comes with a range of holders for practically all applications, including the 3/16” round stem shown here. Pricey, but a great time saver.

![Milling Vise Schematic](image)

**Figure 17 Milling vise schematic**

The inclined surface on the underside of the slide is pocketed for a hemispherical ball, usually hardened. A “pusher block”, with matching inclined surface, propels the slide forward by pressing on the flat surface of the ball. The set screw holds the pusher in loose contact with the ball.
The second main feature of the precision vise is key slots in the base. There are usually two pairs of these, one pair precisely in line with the fixed jaw, the other pair at right angles to it. Key slots can be a great time saver. With snug fitting keys they allow the vise to be removed and replaced routinely, accurately enough for general machining without the need for checking its alignment every time. See "indicating a vise", following page.

Figure 18 Vise keys
This vise came with 16 mm key slots, the same as the T-slots on larger mills. Most bench mills have 12 mm or 14 mm T-slots, calling for necked-down keys as shown inset. These are not commercially available, but can be easily made in the shop.

It is well worth the effort to make vise keys precisely — aim for a snug fit in both vise and table, but not so tight that it takes more than reasonable effort to lift the vise clear.

Figure 19 Movable jaw repositioned for large work
The movable jaw on most precision vises can be installed on the back surface of the slide, greatly increasing the vise's capacity.

Figure 20 Stop assembly
Most milling vises have drilled and tapped holes on the back face for shop-made attachments, in this case a 1/2" ground rod. The stop assembly on the ground rod allows a succession of workpieces to be positioned in exactly the same location every time — one setup, and you're done. This example uses three M5 ratcheting levers for roughing in the setup, plus a brass thumbscrew for final adjustment. Materials: 1 inch square steel bar; 1/4 inch drill rod.
"Indicating" means checking the alignment of the fixed (back) vise jaw relative to the axis of table motion. This is an essential part of everyday machining operations. The following assumes you have not "keyed" the vise, as described earlier.

Install the T-bolts and align the vise by eye. With one of the clamp nuts snug, but not tight, tighten the other one just short of fully-tight (but tight enough so the vise won’t budge without a definite tap from a dead-blow mallet).

A typical setup for indicating is shown below. You need to **make sure the spindle does not rotate** throughout the procedure. If yours is a gearhead machine with no spindle lock, set the gears for the lowest spindle speed. If the mill is belt driven, it may be necessary to lock the spindle in some other way, such as with a wrench on the drawbar nut at the top of the spindle. **Disconnect power before doing this!**

Set the indicator tip against the upper edge of a precision reference bar or, if not available, use the front face of the fixed jaw of the vise instead (check for dings, hone if necessary). Adjust the Y-axis to pre-load the indicator to mid range at the tightly-clamped side of the vise, then lock the Y-axis.

Note the indicator reading, then watch the indicator as you traverse the table slowly toward the loosely clamped side. Ideally, there should be no discrepancy between the indicator readings at the two ends — unlikely at the first attempt. Return the table to the starting point, then repeat the process, tapping the vise in as you go. Repeat the process as often as necessary for the desired accuracy, progressively tightening the "looser" nut. Now fully tighten both nuts, and re-check again (tightening a nut can itself introduce significant error). An established routine like this – tight to loose – can save a lot of time.

There is no “right” setup for a vise, but as a starting point aim for an indicator difference of no more than ± 0.001” over the width of the jaw.

There is no guarantee that a vise with keys, as described on the previous page, will indicate satisfactorily. "Perfect indication" is possible only if the slots in the base of the vise are truly parallel to the fixed jaw. Minor misalignment can often be corrected with a metal shim behind the fixed jaw, as below.

---

**Indicating the vise**
The tip of a standard dial indicator, arrowed, rides along the face of a precision-ground reference bar. Use the vise jaw instead if a reference bar is unavailable.

**Shimming the vise**
Do this if you have installed keys, then found that the vise does not indicated precisely.
ESSENTIAL ACCESSORIES (6) PARALLELS

The workpiece is not always, even often, resting on the floor of the vise. Mostly it needs to be held at some specific height (example: Indicating the Vise photo, previous page). Sets of precision 6” x 1/8” thick “parallels” are available for this, typically covering the range 1/2” to 1-3/4”. Use sponge rubber or other means of keeping front and back parallels against the vise jaws. If the 1/8” parallels get in the way of drills/mills on narrow work, thinner ones are available, e.g., 1/32”, Figure 21 left).

Figure 21 Parallels

ESSENTIAL ACCESSORIES (7) EDGE FINDER

All precise work on the mill is done with reference to a specific edge – usually a vertical edge – on either the machine or the workpiece. There are several types of edge finder, the most popular being the cylindrical style of Figure 22, typical diameters 1/2” and 0.2”. The two cylinders are spring-coupled to each other. Insert the larger diameter into a chuck or collet; then, with the spindle running at high speed, bring the smaller cylinder slowly into contact with the reference edge – this could be the fixed jaw of a vise, or any vertical edge on the workpiece. Expect to see a noticeable wobble of the small cylinder. This will diminish to zero as the edge is approached, followed by a sudden snapping action signifying the precise point of contact. At that point the spindle centerline is exactly 0.1” from the reference edge (or some other value, depending on cylinder diameter). See “X & Y-axis positioning by counting divisions” page 13.

Figure 22 1/2”- 0.2” edge finder
To make the wobble more visible, offset the smaller cylinder before running.

ESSENTIAL ACCESSORIES (8) DRILL BITS

For milling work you will likely need more choices than you can find in the hardware store. This is one of those times when sets are a good idea. In addition to the standard 1/16” to 1/2” set, consider a set of “number” drills, (a.k.a. wire gauge drills) from #80 (0.135”) to #1 (0.228”), and “letter” drills, from A (0.234”) to Z (0.413”).

For the wire gauge set, consider stub length drills instead of the usual jobber length, Figure 23. Because they are much shorter, they are less likely to drift off center, and can often deliver good positioning accuracy without the need for spot-drilling with a center drill.

Be sure you are buying high speed steel (HSS) drills. There are still some carbon steel bits out there.

There are sometimes choices of drill point: 135 degrees is better for hard materials (steel, stainless, etc.) than the standard 118 degrees.

Figure 23 Jobber & stub length drill bits
The twist drill bits here are both 1/4” diameter: (1) Standard (Jobber) length 4” (2) Screw machine length 2-1/2” (3) #3 center drill, 1/4” diameter
Red circles: look for HS or HSS. If not there, beware!

ESSENTIAL ACCESSORIES (9) TAPS

The quality of taps varies widely. They should all be high speed steel (HSS), but carbon steel taps are still available, especially in sets. Don’t be tempted — a broken economy tap is likely a disaster. The two most useful tap variants are:

PLUG (a.k.a. through-hole threading); 3 to 5 of the leading threads are ground in a gentle taper, good general purpose tap for most applications, except tapping blind holes.

BOTTOMING; Also taper ground at the leading end, but only 1 or 2 threads. Better for tapping blind holes, not so good as plug for starting the thread.
What steel is used to make end mills?
Today, end mills are made more or less exclusively of high speed steel (HSS) alloys, some with cobalt to withstand higher temperatures. However, just as with drills, taps and threading dies, if you don’t see HS or HSS on the end mill shank, chances are it isn’t — probably a no-name "carbon steel" instead, which means less wear resistance and a tendency to soften at high temperatures.

Care of end mills
If you can, reserve your new end mills for brass, aluminum and plastics. Use them on steel only when they have lost their initial edge. Finally, when they can no longer do a decent job on steel, send them out for regrinding. There are services that will do an excellent job for less than 1/4 the cost of a new cutter, postage included. The only downside is that the mill no longer cuts its nominal width, not a real issue in most situations.

There are many end mill styles...

The most commonly used tool...
End mills are used for about 80% of all work done on a general purpose milling machine (the other 20% would be drills or special purpose cutters). Whatever the numbers, it’s essential to build a collection of end mills of various sizes and styles.

There are literally thousands of end mills and other cutters in the catalogs — only a tiny fraction of them are shown on the following page. So, the first question might where to begin. The short answer might be to start with just two or three of the "standard diameters", say 1/4", 1/2" and 3/4", then add to them as and when needed. Sets of end mills can be a good buy from time to time, but be sure they are genuine industrial quality.

There are many end mill styles...

The most common are 2-flute and 4-flute. General purpose end mills, for use on practically any material, have a "helix angle" (flute angle) of about 30 degrees relative to the centerline. They are sometimes referred to as "standard spiral" end mills.

Fast spiral end mills, used mostly on aluminum and other soft materials, have a helix angle of a little less than 40 degrees. General purpose 30 degree mills, which can also be used on soft materials, are not as efficient at chip removal as the fast spiral.

Most 2-flute end mills are center cutting, capable of plunge cutting a square-bottomed hole, often without the need to drill a pilot hole. Not all are center cutting, so you need to ask.

Most 4-flute end mills are non-center-cutting. They cannot be used for plunge cutting without a pilot hole larger than the "dead zone" between the radial cutting edges.

Center-cutting 4-flute end mills can be used for plunge cutting without a pilot hole.
Choosing and Using a Mill 3-9-2019.indd

Copyright © 2019 Quality Machine Tools, LLC

1/8” 2-flute standard spiral, double end, 3/16” shank
3/16” 4-flute standard spiral, double end, 3/8” shank
1/4” 2-flute standard spiral, 3/8” shank
1/4” 2-flute fast spiral, 3/8” shank
3/8” 2-flute standard spiral
3/8” 4-flute standard spiral

R8 end mill holder
Use in preference to a collet for most milling with single end cutters

Shop-made fly cutter for 3/8” square tools. With the carbide insert tool shown here, the cutter needs to be run in reverse (counter-clockwise looking down on the workpiece).

Key seat cutter for Woodruff key slots on circular shafts. Cutters usually have a 1/2” diameter shank with radiused neck. This example cuts a 1/16” slot.

1/2” 2-flute standard spiral, double end
1/2” 4-flute 1/2” 2-flute standard spiral
1/2” 2-flute roughing
1/2” 2-flute ball end
5/8” 2-flute standard spiral
3/4” 2-flute standard spiral
1” 4-flute standard spiral, 3/4” shank

Shop-made slitting saw arbors
Use 1/2” diameter mild steel bar or (better) drill rod for the shank. No need for the flat if used with a collet. To maximize saw-cut depth make the hub/clamp plate diameters as small as practicable.

Dovetail cutter
The included angle of dovetail cutters is usually 60 degrees. This is a standard 3/4” x 5/16” cutter, meaning that the angled cut measures 5/16” along the axis. Unlike standard end mills, a dovetail cutter exerts a downward force that may shift its axial position (depth of cut) unexpectedly. Prevent this with a well-lightened collet, or use an end mill holder.

Slitting saws
Saws (1) and (2) are made from sheet HSS material, usually ground concave, thickness range about 0.01” to 3/16”. The more costly style (3) has “side teeth” for better chip removal; thickness range 1/16” to 3/16”. These examples:
(1) 1-3/4” x 0.040” thick, 5/8” hole
(2) 2-1/2” x 0.062” thick, 7/8” hole
(3) 3” x 0.078” thick, 1” hole

Key seat cutter for Woodruff key slots on circular shafts. Cutters usually have a 1/2” diameter shank with radiused neck. This example cuts a 1/16” slot.

Shop-made fly cutter for 3/8” square tools. With the carbide insert tool shown here, the cutter needs to be run in reverse (counter-clockwise looking down on the workpiece).

Key seat cutter for Woodruff key slots on circular shafts. Cutters usually have a 1/2” diameter shank with radiused neck. This example cuts a 1/16” slot.

1/2” 2-flute standard spiral, double end
1/2” 4-flute 1/2” 2-flute standard spiral
1/2” 2-flute roughing
1/2” 2-flute ball end
5/8” 2-flute standard spiral
3/4” 2-flute standard spiral
1” 4-flute standard spiral, 3/4” shank

Shop-made slitting saw arbors
Use 1/2” diameter mild steel bar or (better) drill rod for the shank. No need for the flat if used with a collet. To maximize saw-cut depth make the hub/clamp plate diameters as small as practicable.

Dovetail cutter
The included angle of dovetail cutters is usually 60 degrees. This is a standard 3/4” x 5/16” cutter, meaning that the angled cut measures 5/16” along the axis. Unlike standard end mills, a dovetail cutter exerts a downward force that may shift its axial position (depth of cut) unexpectedly. Prevent this with a well-lightened collet, or use an end mill holder.

Slitting saws
Saws (1) and (2) are made from sheet HSS material, usually ground concave, thickness range about 0.01” to 3/16”. The more costly style (3) has “side teeth” for better chip removal; thickness range 1/16” to 3/16”. These examples:
(1) 1-3/4” x 0.040” thick, 5/8” hole
(2) 2-1/2” x 0.062” thick, 7/8” hole
(3) 3” x 0.078” thick, 1” hole

1/2” 2-flute standard spiral, double end
1/2” 4-flute 1/2” 2-flute standard spiral
1/2” 2-flute roughing
1/2” 2-flute ball end
5/8” 2-flute standard spiral
3/4” 2-flute standard spiral
1” 4-flute standard spiral, 3/4” shank

Shop-made slitting saw arbors
Use 1/2” diameter mild steel bar or (better) drill rod for the shank. No need for the flat if used with a collet. To maximize saw-cut depth make the hub/clamp plate diameters as small as practicable.

Dovetail cutter
The included angle of dovetail cutters is usually 60 degrees. This is a standard 3/4” x 5/16” cutter, meaning that the angled cut measures 5/16” along the axis. Unlike standard end mills, a dovetail cutter exerts a downward force that may shift its axial position (depth of cut) unexpectedly. Prevent this with a well-lightened collet, or use an end mill holder.

Slitting saws
Saws (1) and (2) are made from sheet HSS material, usually ground concave, thickness range about 0.01” to 3/16”. The more costly style (3) has “side teeth” for better chip removal; thickness range 1/16” to 3/16”. These examples:
(1) 1-3/4” x 0.040” thick, 5/8” hole
(2) 2-1/2” x 0.062” thick, 7/8” hole
(3) 3” x 0.078” thick, 1” hole

Key seat cutter for Woodruff key slots on circular shafts. Cutters usually have a 1/2” diameter shank with radiused neck. This example cuts a 1/16” slot.

Shop-made fly cutter for 3/8” square tools. With the carbide insert tool shown here, the cutter needs to be run in reverse (counter-clockwise looking down on the workpiece).
X & Y-axis positioning by counting divisions

For all spindle positioning operations, with or without DROs, avoid using the quill lock. Retract the quill fully, then adjust the Z axis instead.

Why? On most vertical mills, including the heavier knee mills, locking the quill may offset the spindle by one or two thousandths of an inch. If the edge of the workpiece has been “found” in the quill-locked condition, this will affect placement of holes drilled thereafter. Instead, lower the quill with the fine downfeed control. This is worm driven, so it stays where it’s put without locking.

NOTE: The above does not apply to operations calling for precise depth control, such as milling. For such operations the quill must be locked to maintain a given depth of cut.

The following assumes a table motion of 0.1” per turn of the handwheels — yours may differ.

Figure 25 is an example of Y-axis positioning. A hole is to be drilled 0.25” on the Y-axis relative to the front edge of a workpiece in a vise, or otherwise clamped to the table.

1. Install an edge-finder in collet or chuck (a tip diameter of 0.2” is assumed).
2. Lock the X-axis by tightening both levers.
3. If the reference edge is already to the back the spindle centerline, do nothing; if not, rotate the Y-axis handwheel clockwise to send the workpiece backwards (toward the column).
4. Engage the fine downfeed.
5. With the spindle running, lower the quill as necessary using the fine downfeed handwheel; bring the table forward (counter-clockwise), stopping at the point where the edge-finder just makes contact (the tip jumps out of line). Stop the spindle.
6. While holding the Y-axis handwheel to prevent movement, zero the dial, then re-tighten the dial thumbscrew.
7. Raise the quill, then rotate the handwheel one exact full turn counter-clockwise (0.1”) to bring the reference edge to the spindle centerline.
8. Rotate the handwheel 2-1/2 turns counter-clockwise to bring 50 on the dial opposite the datum; the spindle is now exactly 0.25” behind the reference edge.

Tilting the headstock

In routine operations the user relies on squareness of the spindle relative to both axes of the table. Front-to-back squareness set at the factory, and is not adjustable (by everyday methods), but in the other plane the headstock can be set to any angle up to 90 degrees either side of the normal vertical position. [Because re-establishing true vertical – tramming — on any mill is a time consuming process, look first for other ways of handling the project instead of tilting the head.]

The headstock is usually secured by three nuts spaced 120 degrees apart, one underneath and one either side. The headstock is top-heavy, and may swing suddenly to either side unless a helper is on hand to restrain it.

Set the headstock to the desired angle by reference to the tilt scale, then re-tighten the nuts. Bear in mind that the scale is good only to approximately ± 0.25°, so a more accurate means of angle measurement will be needed if the project calls for precise tilting.

Tramming the headstock

Tramming means accurate alignment — in this case adjusting the headstock tilt to bring the spindle to a known angle — usually 90 degrees — relative to the table.

As shipped, the mill is set to zero tilt, squared accurately enough for out-of-the-box test drillings, etc. For more demanding project work thereafter, the spindle needs to be set at precisely 90 degrees, in other words trammed. “Out of tram” may show up as an offset of a few thousandths between entry and exit of a deep hole, or as a scalloped effect when surfacing a workpiece, Figure 26.

Tramming is done by fine-tuning the headstock tilt angle. Tram is typically checked by attaching a dial indicator to some form of “sweepable” holder installed in the spindle, the aim being to adjust tilt for the same reading on either side of the X axis.
Figure 27  Shop-made sweeping holder for dial indicator
This example shows a rectangular section aluminum bar with threaded holes allowing the choice of two sweep diameters, 6” and 10”, measured from spindle centerline to indicator tip. The smaller sweep can be used for front-to-back tramming, also left-to-right as here. For more sensitive left-to-right tramming, use the larger sweep.

Suggested procedure for tramming
Figure 27 shows a typical shop-made holder; it has a threaded arbor allowing the choice of two radius arms, 6 and 10 inches measured from spindle centerline to indicator tip. A collet is used to hold the arbor, in this case 5/8” diameter. The dimensions are arbitrary, but the indicator must be firmly attached, and the arm rock-solid relative to the indicator spring force (which can be considerable on plunger-type indicators).

A suggested procedure for re-establishing tram:

1. Disconnect power.
2. Set the headstock to the approximate zero degree position on the tilt scale, then tighten the nuts enough to avoid unexpected headstock movement.
3. Remove the vise and clean the table surface.
4. Set a 1-2-3 block (or other precision-ground block) on the table under the indicator probe.
5. Switch on the quill DRO, if available.
6. Lower the spindle using fine downfeed to give a half-scale indicator reading (exact location isn’t important, but remember the reading).
7. Note both the dial indicator and DRO readings, then back off the fine downfeed a couple of turns to avoid collision when sweeping.
8. Select the highest spindle speed (this allows you to sweep the indicator holder easier from side to side).
9. Reposition the 1-2-3 block to the opposite location on the table.
10. Swing the indicator holder to the new location, then lower the spindle – fine downfeed again – to give the same dial indicator reading as in step (7).

If the headstock is perfectly trammed – highly unlikely at the first shot – the DRO reading should be as in step (7). If not, loosen the nuts just enough to allow the headstock to be tapped a fraction of a degree in the direction called for, then re-tighten the nuts. (The “tap” can be anything from a gentle hand-slap to a rap with a soft-face dead-blow mallet).

Repeat steps (4) through (10) until satisfied with the tram, tightening the nuts as you go. This will likely call for several iterations. There is no “right” tram; the acceptable difference in side-to-side readings depends on project specs. As a starting point, aim for ± 0.001” on a radius of 5 or 6 inches.

A similar procedure may be used to check tram in the Y-axis, front to back. The difference here is that there is that Y-axis tram is established in manufacture, and can be adjusted only by shimming the column-to-base interface. This is a two-person procedure, possibly requiring an engine hoist or some other means of un-weighting the column and headstock. Bear in mind that the front-to-back tilt can vary as the headstock is raised and lowered.

Tramming calls for patience on any mill! Expect to tighten and re-check at least three times (simply tightening the bolts can affect the tram).
NEW TO ALL THIS?

Here are some more pointers that may be helpful.

**Milling feeds and speeds**

What spindle speed should I use? What cutter size? What depth of cut? How fast to move the table? In production operations, all these things are well known — they are optimized for the best compromise between throughput and tool life. In the model shop, though, we have a quite different situation. For one thing, most every work session is different from the one before. For another, we don’t usually have reliable knowledge of cutter quality and workpiece material. Finally, looking at weight, rigidity, and speed range, there’s no real comparison between a 1 HP model shop mill and its 10 HP industrial counterpart.

Spindle speed is the key factor. Cutter manufacturers will recommend a “cutting speed”, in feet per minute, but that’s not directly helpful (cutting speed means speed at the periphery of the tool, a function of both spindle speed and cutter diameter). So the best we can do instead is work from a simplified table of spindle speeds for various materials and cutter sizes. If you really are new to all this, bear in mind that the most frequent source of problems is small cutters running too slow, and large cutters too fast.

<table>
<thead>
<tr>
<th>Cutter Diameter</th>
<th>Cutting Speed (ft/min)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1/4”</td>
</tr>
<tr>
<td>O1 tool steel</td>
<td>50</td>
</tr>
<tr>
<td>Cast iron</td>
<td>60</td>
</tr>
<tr>
<td>W1 tool steel</td>
<td>85</td>
</tr>
<tr>
<td>Mild steel</td>
<td>100</td>
</tr>
<tr>
<td>Brass/aluminum</td>
<td>250</td>
</tr>
<tr>
<td></td>
<td>3/8”</td>
</tr>
<tr>
<td>O1 tool steel</td>
<td>750</td>
</tr>
<tr>
<td>Cast iron</td>
<td>900</td>
</tr>
<tr>
<td>W1 tool steel</td>
<td>1300</td>
</tr>
<tr>
<td>Mild steel</td>
<td>1500</td>
</tr>
<tr>
<td>Brass/aluminum</td>
<td>3750</td>
</tr>
<tr>
<td></td>
<td>1/2”</td>
</tr>
<tr>
<td>O1 tool steel</td>
<td>500</td>
</tr>
<tr>
<td>Cast iron</td>
<td>600</td>
</tr>
<tr>
<td>W1 tool steel</td>
<td>850</td>
</tr>
<tr>
<td>Mild steel</td>
<td>1000</td>
</tr>
<tr>
<td>Brass/aluminum</td>
<td>2500</td>
</tr>
<tr>
<td></td>
<td>3/4”</td>
</tr>
<tr>
<td>O1 tool steel</td>
<td>400</td>
</tr>
<tr>
<td>Cast iron</td>
<td>450</td>
</tr>
<tr>
<td>W1 tool steel</td>
<td>650</td>
</tr>
<tr>
<td>Mild steel</td>
<td>750</td>
</tr>
<tr>
<td>Brass/aluminum</td>
<td>1250</td>
</tr>
</tbody>
</table>

Suggested spindle speeds for HSS mills

1. Carbide mills should be run faster, between X2 and X4.
2. Mild steel means any low carbon steel (1018) and leaded alloys (12L14).
3. These are only starting figures — workpiece hardness and cutter quality/sharpness vary widely.
4. Most mills do not offer these exact speeds, especially the highest speeds suggested for brass and aluminum — use the nearest available.

Which way to move the table?

This depends on two factors: (1) Which face of the workpiece is being machined — front, back, left or right, and: (2) Whether you are milling conventionally, or climb milling.

In conventional milling (a.k.a. Up Milling) the workpiece is fed against the cutter’s rotation, Figure 28. Climb milling (Down Milling) has the workpiece going in the same direction as the cutter. Unless drag is applied to the table, the cutter can itself apply traction to the workpiece, causing unexpected motion and — potentially — damage to the cutter. If there’s a choice, DO NOT climb mill.

Depth of cut

Assuming the spindle speed is correct for the material and cutter diameter (see the table), depth of cut is usually arrived at by experiment. Always start on the light side, even as little as 0.02”, with a slow feed rate on the table. Increase both cut and feed in small increments, stopping at the point where the machine begins to vibrate noticeably.

Don’t cut slots with a 4-flute cutter

Why? Because they cut oversize, and leave the sides of the slot rough. If the slot width is to be (say) 5/16”, use a reground (smaller) 5/16” to pre-cut the slot, then make a second pass to size. (Or, you can stay with the undersize cutter, making two finishing passes, one left, one right.) This problem doesn’t arise with 2-flute cutters — one reason they’re sometimes called slot drills.
**Drilling & reaming**

All drills cut oversize holes, ranging from about 0.002" for a 1/16" diameter drill to about 0.005" for 1/2" diameter. This applies even to new drills.

Whenever possible use stub length drills instead of the standard jobber length. They stay on center better and, because they don’t flex so much radially, drill a more round hole. For precise work, “spot drill” each hole location using a center drill before drilling to size.

If you need an exact hole size, drill a few thousandths undersize then follow with a reamer (or, if the hole is large enough, bore to size). Reamers are expensive, so consider buying individual sizes as needed instead of sets. Do not be tempted by adjustable reamers. They sound good — but, like broken-tap extractors, have never been known to work well. Run reamers about 1/4 the speed of the corresponding drill size, using plenty of cutting oil. Never run them in reverse.

If a reamer isn’t available, drill a few thousandths undersize as before, then re-drill with a brand new drill of the nominal hole size, using very little downward pressure.

**Choice of steel**

If you have the choice, always go with 12L14 or other free machining steels, usually alloyed with lead. 12L14 machines faster, more cleanly than off the shelf steels such as 1018. Its only downside is that it is available in only 3 standard sections: square, round, and hexagonal.

**Lubricants for steel cutting**

For drilling and tapping, use one of the many cutting fluids such as Tapmatic, Tapmagic, Relton, etc. They can, and should, be used on all steels (including stainless) and also on aluminum. For threading steel, especially larger diameters, most users find Castrol’s Moly Dee the most reliable fluid.

For routine milling (and turning) consider using a “water miscible coolant” such as Blaser Swisslube. When mixed ready for use this is a white lightweight fluid that handles just like water. It is much less messy than any neat oil product, and can be applied using a spray bottle. Cleanup with paper towels. Although it is mostly water, it does not increase the likelihood of rusting on the mill, *provided it has been mixed* according to directions.

Instead of buying the unmixed product — large container, expensive — low volume users may be able to buy a gallon or two of ready mixed product from a local machine shop.

**Thread tapping**

When threading a drilled hole it is essential to align the threading tap properly in the bore. The mill is often used for this purpose, ideally with a dedicated tap holder or, for production work, an auto-reverse tapping attachment. Typically, all you need do for sizes up to M6 or 1/4” is swap the drill for the tap, then hand-turn the chuck while keeping a gentle downward pressure on the quill. After 2 or 3 turns, the tap may be stable enough for completion using a hand wrench. Tapping can also be done under power, which may be kinder on the tap than turning by hand — but this is a judgment call based on experience. For either method, it is essential to use a *tapping fluid*. Any cutting oil is better than none, but it’s worth repeating that Castrol’s Moly Dee is the most reliable for threading steel.

If power-tapping bear in mind that spindle reverse is not instantaneous, so be careful tapping blind holes. Be sure the quill locking lever is free, and start trial work with the lowest spindle speed.