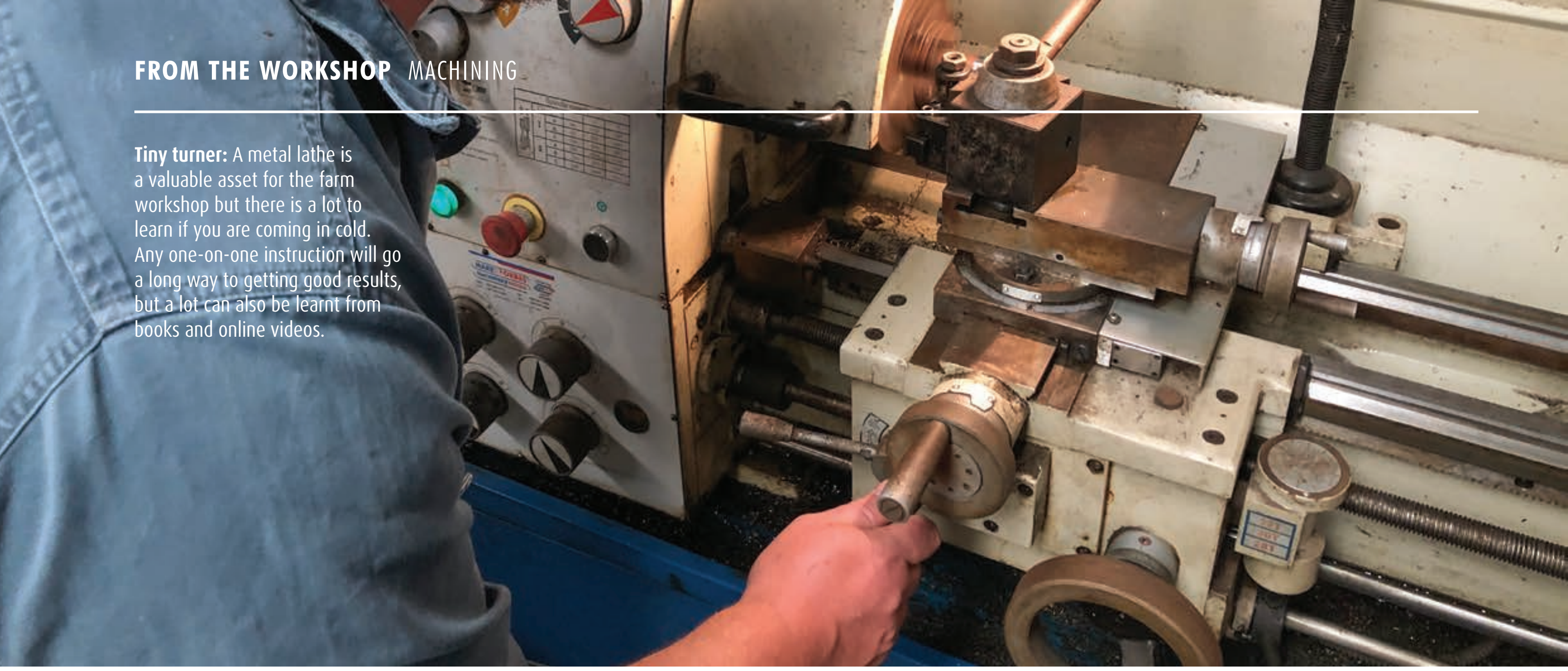


Tiny turner: A metal lathe is a valuable asset for the farm workshop but there is a lot to learn if you are coming in cold. Any one-on-one instruction will go a long way to getting good results, but a lot can also be learnt from books and online videos.

Basic guide for metal lathes

A metal lathe is probably the most versatile tool you can add to the farm workshop, and is only limited by the skill of the operator and tooling available. By Josh Giumelli

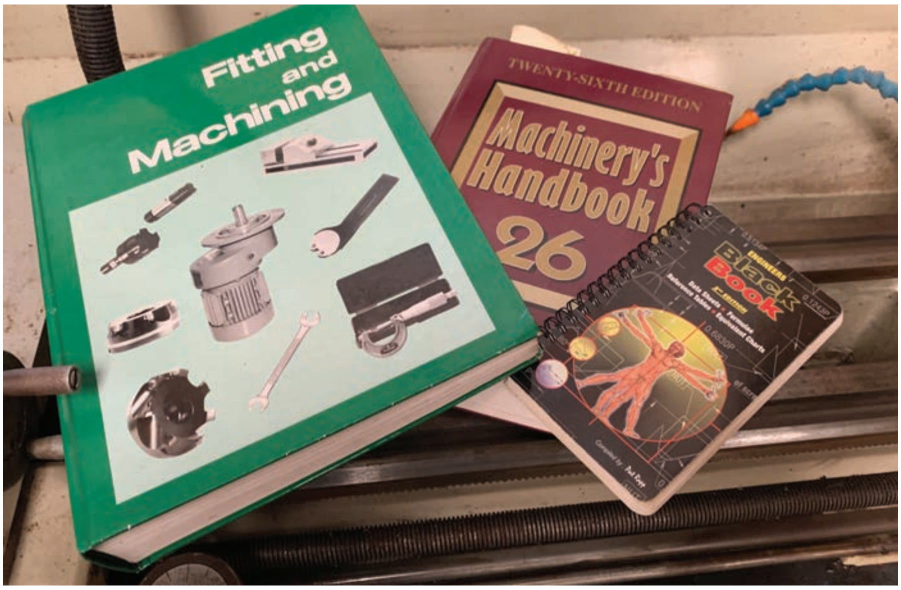
In last month's *Workshop*, we covered what to look for when buying a lathe for the farm, and what accessories should be included in the purchase. This month, we look at the basic functions of the lathe, how to use the various accessories, and what tooling you should acquire.

There must be a strong focus on safety whenever operating a machine tool such as a metal lathe. Rotating spindles can entangle limbs, leading to severe injury or death.

A metal lathe has significant torque and will not simply stall if you get caught. Treat the spindle with the same respect you would an unguarded PTO shaft.

Metal turnings or swarf is ejected from the cutting tool at speed, often towards the operator. It is often hot and usually razor-sharp and should never be touched with bare hands. It can also hook in loose clothing, dragging it into the rotating spindle. Small swarf chips also pose a significant risk to the eyes and face, so always operate a lathe with eye protection or full-face shield.

There is much to learn when starting out with machine work; far more than can be covered in this article. But knowledge and skill can be acquired through instruction, reading books, watching online videos and simply getting stuck in and having a go. Rather than a comprehensive tutorial on lathe operations, this article serves as an introduction to using a metal lathe, and the range of tasks it can handle.

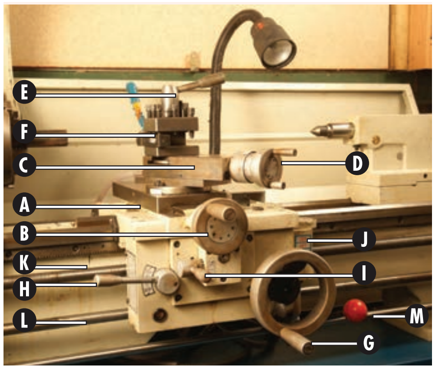


Ready reference: A couple of reference books which will be handy for anyone learning to use the metal lathe include: Fitting and Machining ($99), a 640 page reference book used in TAFE colleges across Australia; Machinery's Handbook (about $200), a 3000 page bible; Engineer's Black Book ($77), a 235 page pocket handbook of all things mechanical and machining.

THE CARRIAGE

The lathe carriage is used to manipulate the tool by either moving it parallel to the lathe bed for turning and boring, or across the bed for facing (see Figure 1.) The carriage handwheel (pictured) moves the carriage rapidly along the bed and is more for repositioning the tool rather than manipulating it in a cut. The cross slide moves the tool across the face of the work at right angles to the lathe bed and has a micrometer dial for accurate positioning. The carriage can be made to move along the bed automatically using automatic feed, or the cross slide can be operated on auto feed for facing operations.

Figure 1. The lathe carriage



A: Cross slide; B: Cross slide hand wheel; C: Compound cross slide; D: Compound cross slide hand wheel; E: Tool post; F: Cutting tool; G: Carriage hand wheel; H: Split nut lever (engages lead screw); I: Automatic feed lever; J: Threading dial; K: Lead screw; L: Automatic feed driveshaft; M: On/off lever

The compound cross slide can be rotated 360 degrees on its base, and is used for cutting tapers and bevels, controlling the cut depth when cutting threads, or simply to fine tune the tool position, bringing it into contact with the work. It cannot be operated automatically. Note the micrometer dial for accurate measurement.

THE TAILSTOCK

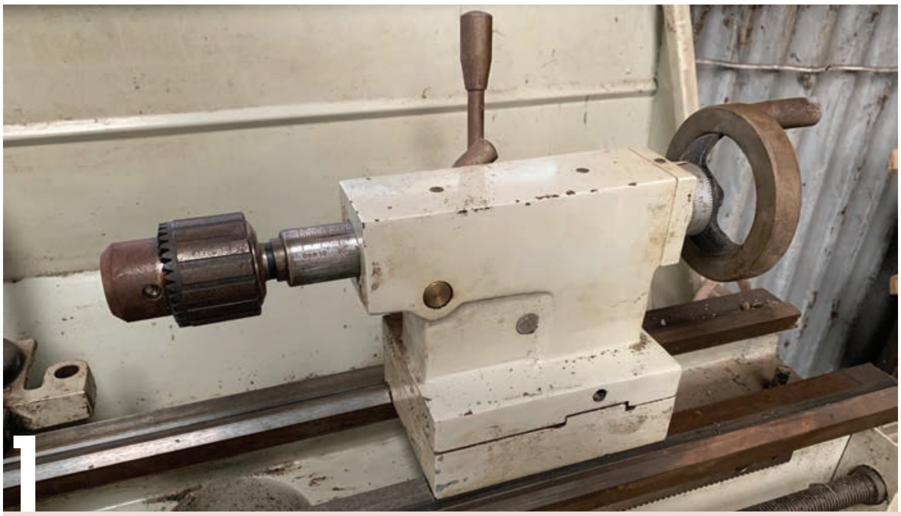
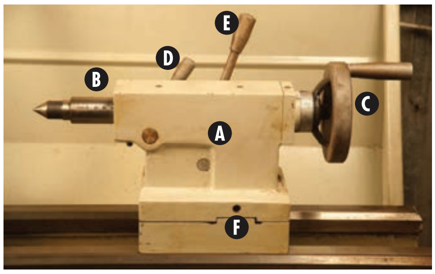


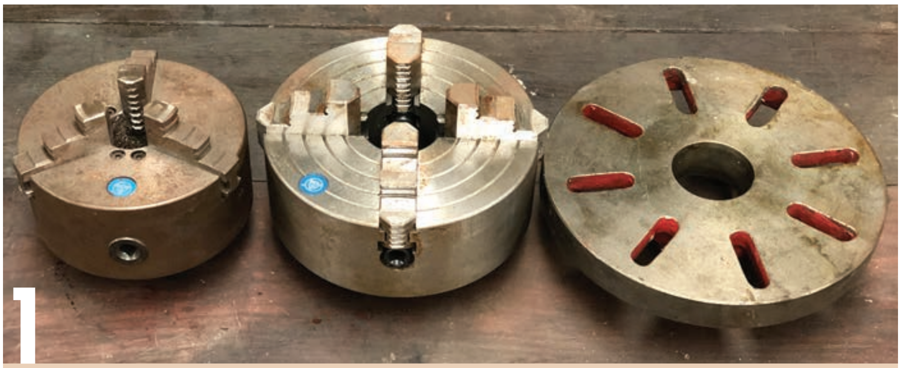
Figure 2. The tailstock



A: Tailstock body; B: Tailstock quill; C: Quill feed hand wheel; D: Quill lock lever; E: Tailstock lock lever; F: Tailstock set-over screw (for alignment and taper turning)

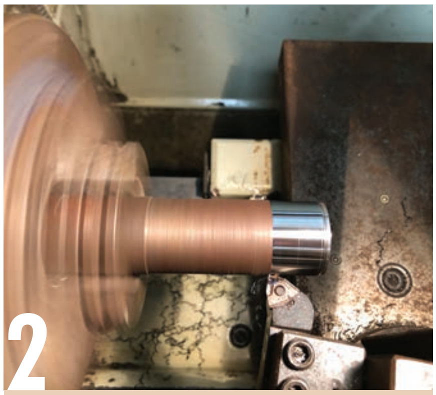
The tailstock is used for supporting long work or holding a drill bit when drilling holes. The tailstock is slid along the bed and locked into position with the locking lever. The tailstock ram or quill is hollow and features a Morse tapered socket for holding attachments. The handwheel is used to advance the tailstock ram and has a micrometer collar for accurate depth control. The ram can be locked in place with the short lever. If the ram is retracted fully it will eject the attachment from the ram socket.

WORK-HOLDING



1

The main methods for mounting work in a lathe are in a three-jaw chuck (left), a four-jaw chuck (centre) or on a faceplate (right). The three-jaw chuck is self-centering and is used to grip cylindrical work, like a drill chuck. The four-jaw chuck features individually adjustable jaws and is used to hold irregularly-shaped work, although there are self-centering versions available. The faceplate is ideal where it is more convenient to simply bolt the work to the lathe using any existing holes in the work. See last month's article for more information of work-holding and fixed and traveling steadies.

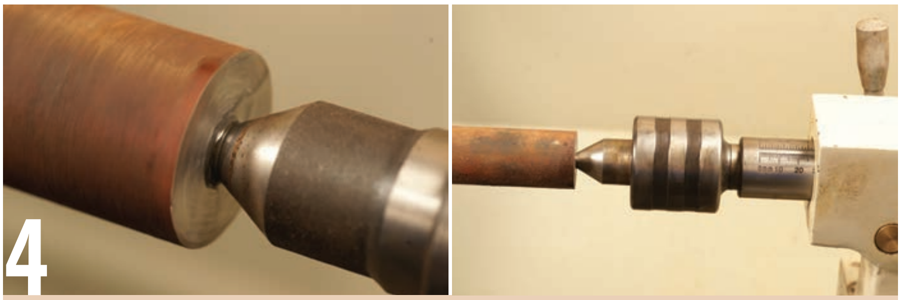


2

Short lengths of cylindrical stock are simply clamped in the three-jaw chuck for turning and facing operations. Due to the short length and large diameter, the work will not flex under pressure from the tool.

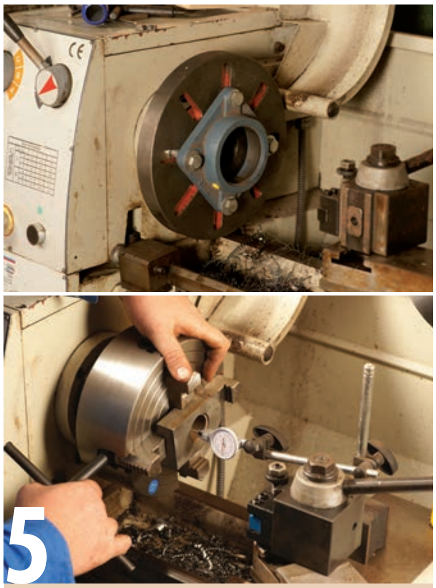
3

Longer work needs to be supported by a tailstock centre. The shaft centre must be marked and then centre-drilled to create a countersunk hole which will engage with the tailstock centre. The centre is advanced into the work by turning the tailstock handwheel.



4

The two most commonly used tailstock centres are the dead or plain centre (left) and the live centre (right). The dead centre requires a drop of oil periodically to keep it lubricated and cool. The live centre is equipped with bearings and rotates with the work. If the point is not rotating when the spindle is turning, increase the pressure on the centre by advancing the tailstock ram slightly.



5

Work held in the faceplate (top) or four-jaw chuck (bottom) often requires aligning, such as when an existing bore needs to be centered on the axis. A dial test indicator (DTI) is mounted to the lathe bed using a magnetic base, and the spindle slowly rotated by hand, noting the run-out on the DTI. The position of the work can then be adjusted to zero-in the run-out. Any work with a simple centre-mark can be simply aligned with a centre in the tailstock.

TOOLING

1

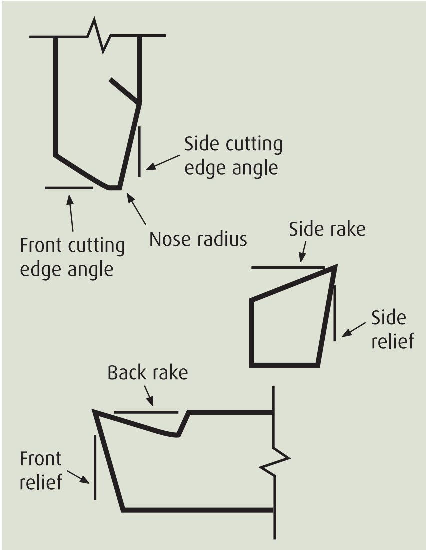
Lathe tooling is available in a large range of shapes and sizes. The plain turning tools shown left show the three main types; high speed steel (HSS, left), cemented carbide tip (centre), and indexable carbide insert (right). HSS is available in blanks in a range of sizes and is ground to shape. It is the cheapest form of tooling but does require skill to grind to shape. Cemented carbide tip tools are more expensive and harder than HSS, but still require grinding to maintain a sharp edge. Indexable insert tools are by far the most expensive choice and are the mainstay of the manufacturing industry.

2

HSS tool blanks are ground to suit a range of turning tasks and materials but must be kept cool during grinding to prevent softening. Use a dressed grinding wheel, preferably a green silicon carbide wheel which is sufficiently hard enough for the task. For extra-sharp tooling, blanks can be finished off on an oilstone. See table 1 for grinding recommendations.

Table 1. Lathe Tools – HSS angles (degrees)

Material	Side Relief	Front Relief	Side Rake	Back Rake
Aluminum	12	8	15	35
Brass	10	8	5 to -4	0
Bronze	10	8	5 to -4	0
Cast iron	10	8	12	5
Copper	12	10	20	16
Machine Steel	10 to 12	8	12 to 18	8 to 15
Tool Steel	10	8	12	8
Stainless Steel	10	8	15 to 20	8



HSS tools can be sharpened and reshaped countless times before they are too small for further use. Tools can also be custom ground for specific jobs such as cutting chamfers, radii, circlip or O-ring grooves (pictured).

Indexable insert tools can be purchased individually, or in sets, which are often more cost-effective. The inserts are not sharpened when they are blunt, but are simply removed from the tool and rotated (or indexed) to use a fresh cutting edge. When all edges are blunt or damaged, the tip is discarded an a new one fitted. Tips are available in a wide range of types and typically cost around $10 each.

TURNING

Turning involves reducing the diameter of a part by running a cutting tool along its length. The cutting tool is advanced into the work using the cross slide to take progressively deeper cuts until the required diameter is achieved. The tool is moved along the lathe bed using the automatic feed, the compound cross slide, or the carriage hand wheel.

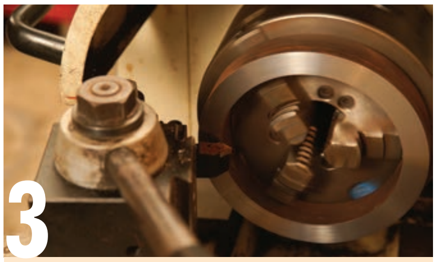
The tool can travel either left to right, or right to left and there are separate left or right-handed cutting tools to suit. Most turning is performed from right to left as this is the default travel direction on auto-feed systems.

To turn down a shaft, set the spindle in motion and advance the tool until the tip just contacts the work. Set the micrometer sleeve on the cross slide handwheel to zero (or zero the digital readout unit). Advance the tool into the work the desired depth and engage the auto feed (right). Note that one complete turn of the handwheel on this lathe advances the tool 2mm into the work, but the scale actually registers 4mm, as this is the amount removed from the **diameter** of the work. See Tables 2, 3 and 4 for recommended cut depths, spindle speeds and feed rates.

FACING

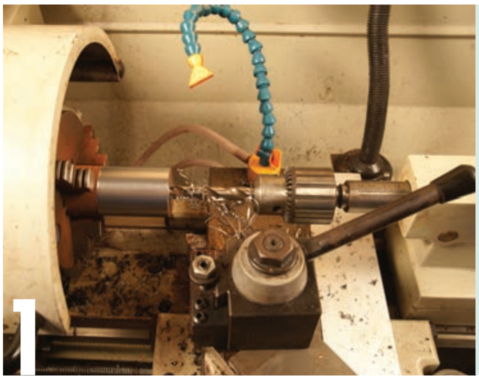
Facing is performed by advancing the tool into the work, across the face perpendicular to the lathe bed, using the cross slide. Facing operations are used to machine the end of a shaft flat (pictured), re-surface a disc brake or flywheel or flatten the surface of component, such as a bearing mount. Note that items do not have to be circular, in fact any component that can be held in chuck or on a faceplate can be machined flat.

Facing tools are similar to turning tools but are orientated to allow feed at right angles to the bed. Facing tools can be HSS, tungsten carbide tipped or indexable insert.



Roughing face cuts should travel from the outside in towards the centre, but fine finishing cuts can be performed in either direction. Full facing cuts should be stopped as they pass the centre of the workpiece. On most lathes the auto feed can be selected for facing, and standard direction is in towards the centre.

DRILLING



Drilling is a common process in the metal lathe and involves feeding a stationary drill bit towards a rotating workpiece, opposite to that performed in a drill press. Speeds and feeds for drilling are the same as would be used for a drill press.

All drilling operations should start with a centre drill held in the tailstock. Normal drill bits will tend to wander badly unless the work is heavily centre-punched.

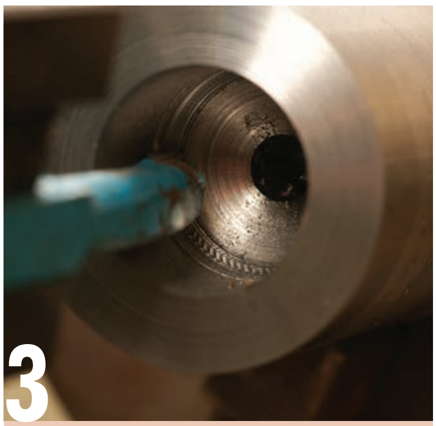
Small drill bits can be held in a tailstock chuck, which is simply a drill chuck with a Morse tapered shank which fits in the tailstock ram. In some cases a tapered shank chuck can be used from the workshop drill press. Adapter sleeves are available allowing a smaller taper to fit into a larger taper.

The other option is to use a tapered shank drill bit in place of the chuck. This is held directly in the tailstock ram socket. For drilling, the tailstock is slid along the bed and locked in position, then the drill bit is advanced into the work using the tailstock handwheel. Make sure the drill is withdrawn often on deep holes to help clear swarth which will become jammed in the flutes.

BORING

Boring is analogous to turning, except on the internal surface rather than the external. Boring is simply another form of drilling, but usually on a larger scale. Boring allows any choice of internal diameter, rather than drilling which is limited to the size of the drill bit used. Boring is fed in from the end of the cylinder by a boring bar held in the toolpost.

Boring bars come in many forms and can be fully-forged from a single section of HSS. The most common types are HSS (shown left installed in a boring bar), carbide-tipped (centre) and indexable insert (right). The challenge is to allow enough tool overhang to reach the full depth of the bore without having undue flex in the tool which would cause chatter and spoil the surface finish. Boring tools for small diameter holes need sufficient clearance to allow the cutting edge to contact the surface without the rest of the tool holder fouling on the work.



Keep a close eye on the tool progress so that it doesn't bottom out on the base of the bore, or the lathe chuck. If you require a flat base to the bore there are internal facing tools available.

THREADING

Cutting threads is one of the more complex lathe operations as it involves a large range of variables. In most cases, other threading options such as dies and taps are used in preference to cutting a thread on the lathe due to the time taken to set up the lathe.

External threading tools (left) have a single point which is ground to a 60 degrees angle for metric and Unified National Coarse and Fine (UNC and UNF) threads, or 55 degrees for Whitworth or British Standard Fine (BSF) threads. Threading tools are available in tipped carbide or indexable inserts or can be ground from HSS. Internal threading tools (right) resemble boring bars and can suffer from the same rigidity issues if used with heavy cuts and excessive overhang.

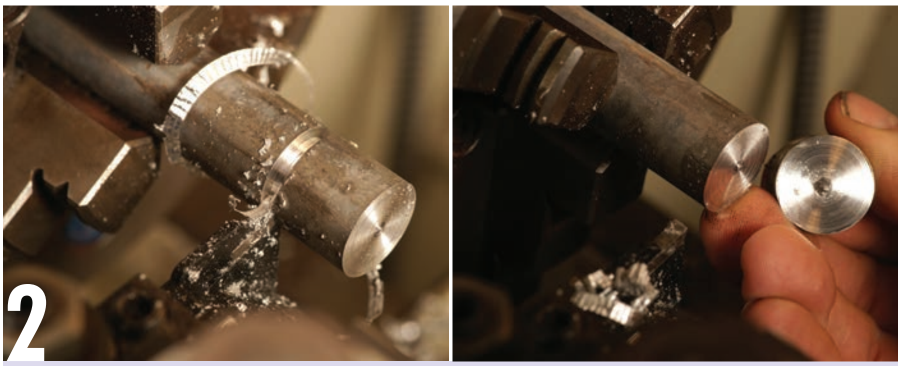
Threading begins with turning stock down to the major thread diameter for external threads (as above), or boring to the minor thread diameter for internal threads. Threading tools are only used to cut the actual thread forms, not to reduce the diameter of the stock from which the thread is cut.

Thread pitch, or the distance between threads, is governed by the amount the carriage moves for each turn of the spindle. Larger lathes will have a thread cutting gearbox, which is used to select the specific lead screw ratio for the thread pitch required (left). Smaller lathes or older units may use change gears to select the leadscrew ratio instead (right). The lathe should have a thread-cutting chart which is essential for correct setup. Note that it is possible to cut a metric thread on an imperial lathe, and vice versa, provided the transposing gear is used in the leadscrew drivetrain.

Threading is done at slow spindle speeds and with several passes of the tool. The leadscrew is engaged with the half-nut lever, which must be re-engaged in the same spot for each pass. Alternatively the spindle can be reversed to the start of the thread.

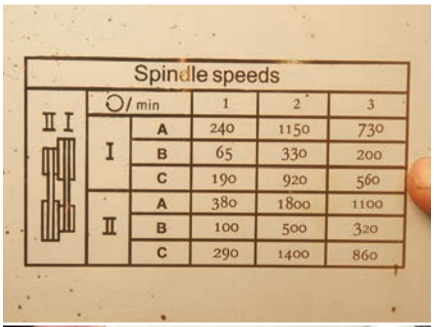
PARTING OFF

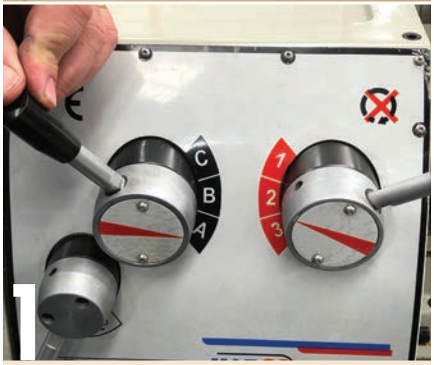
Parting off is the process of cutting off a section of stock from the workpiece, generally when turning operations have been completed. Shown above are a HSS parting blade in holder (left), two carbide tipped parting tools (centre) and replaceable insert parting tool (right).



Parting off is one of the more difficult lathe operations. Ensure the tool is as rigid as possible and set it to centre height. Use plenty of lubrication in the cut and apply with firm pressure on the cross slide handle. Use a lower spindle speed, maintaining cutting speeds of 11m/min for mild steel, or 17m/min for brass. Take care when the cut is almost complete as the offcut will likely snap leaving a small nib in the centre of the stock.

FEEDS AND SPEEDS





Spindle speeds, cutting depth and feed rates are important variables to control when undertaking any turning operation. It is best to aim on the slow side, with low spindle speeds, shallow cuts and low feed rates as you get used to using a lathe. This is especially true on an older, worn lathe. Modern lathes use a spindle drive gearbox, with speed selected from a chart similar to that shown above. Spindle speed depends on the diameter of the work, as this will influence cutting speed. Cutting speed is a surface speed and is measured in metres per minute (m/min) or feet per minute in the imperial system. Different materials will have different optimum cutting speeds, so a larger diameter item will need to turn more slowly than a smaller item to keep the m/min within spec. See Table 2 for recommended cutting speeds for different materials. Table 3 can be used to quickly calculate lathe spindle speed based on work diameter and cutting speed, or the following formula can be used:

Spindle rpm = (320 x cutting speed)/diameter

Where cutting speed is in m/min and diameter is in mm.

Table 2. Cutting speed for common materials

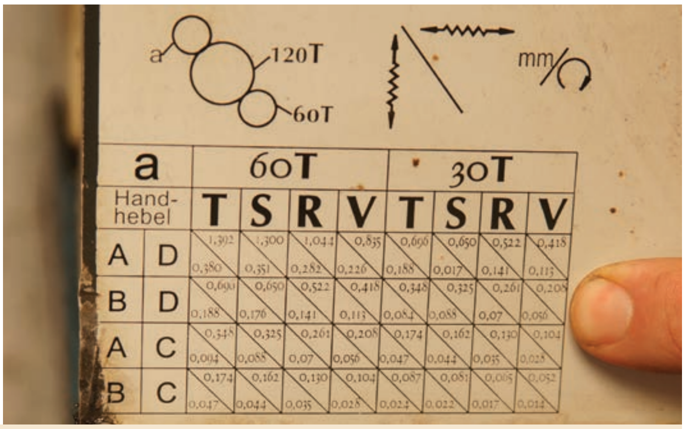
Material	Cutting speed (m/min)
Aluminium	75-105
Brass	45-60
Bronze	24-45
Cast iron	18-24
Stainless steel	23-40
Steel (free machining)	35-69
Steel (mild)	30-38
Steel (tough)	15-18

Table 4. Feed rates and cut depth for mild steel

Cut depth (mm)	Cutting speed (m/min)	Feed (mm/rev)
0.8	65	0.4
	85	0.2
1.5	36	0.8
	53	0.4
	67	0.2
3.2	21	1.6
	30	0.8
	42	0.4
	54	0.2

Table 3. Lathe spindle speeds for cutting speed

Surface speed	Work diameter (mm)												
m/min	10	20	30	40	50	60	75	100	125	150	200	250	300
10	318	159	106	80	64	53	42	32	25	21	16	13	11
15	477	239	159	119	95	80	64	48	38	32	24	19	16
20	637	318	212	159	127	106	85	64	51	42	32	25	21
25	796	398	265	199	159	133	106	80	64	53	40	32	27
30	955	477	318	239	191	159	127	95	76	64	48	38	32
35	1114	557	371	278	223	186	149	111	89	74	56	45	37
40	1273	637	424	318	255	212	170	127	102	85	64	51	42
45	1432	716	477	358	286	239	191	143	115	95	72	57	48
50	1591	796	530	398	318	265	212	159	127	106	80	64	53
55	1750	875	583	438	350	292	233	175	140	117	88	70	58
60	1910	955	637	477	382	318	255	191	153	127	95	76	64
65	2069	1034	690	517	414	345	276	207	165	138	103	83	69
70	2228	1114	743	557	446	371	297	223	178	149	111	89	74
75	2387	1194	796	597	477	398	318	239	191	159	119	95	80
80	2546	1273	849	637	509	424	339	255	204	170	127	102	85
85	2705	1353	902	676	541	451	361	271	216	180	135	108	90
90	2864	1432	955	716	573	477	382	286	229	191	143	115	95
95	3024	1512	1008	756	605	504	403	302	242	202	151	121	101
100	3183	1591	1061	796	637	530	424	318	255	212	159	127	106
105	3342	1671	1114	835	668	557	446	334	267	223	167	134	111
110	3501	1750	1167	875	700	583	467	350	280	233	175	140	117



The feed rate of the cutting tool is another important consideration. Feed rate is defined as the amount the tool moves into the work per revolution of the lathe spindle, measured in millimetres (see table 4). An overly high feed rate will cause excessive heat, leave a poor surface finish and may damage the cutting tool. But depth of cut is equally important, as an overly deep cut will cause the same issues. Metal lathes should have a feed rate table attached to the headstock, which is selected using a gearbox (often the threading gearbox), or change gears, or a combination of both (as is the case above). This is the rate at which the tool moves when the automatic feed lever is engaged on the carriage (left). High feed rates and cut depths are generally used when roughing parts down to size, then reduced for accurate sizing to the final dimension with improved surface finish. Faster spindle speeds at low feeds will improve the surface finish.