1.3 LATHE:

Lathes are machines which are used in the fabrication of parts which are symmetrical about an axis of rotation, such as shafts, pins, threaded components etc.

Lathes have tools mounted on a component of the machine which can move in the x and y directions, while the work-piece, held securely in a chuck, rotates.

The process of producing such an axi-symmetric part is called “turning”.

Figures 1.3.1 and 1.3.2 show typical lathes:

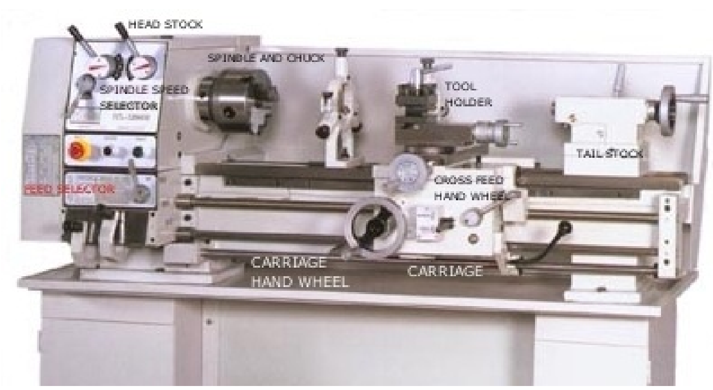

Figure 1.3.1: Engine Lathe

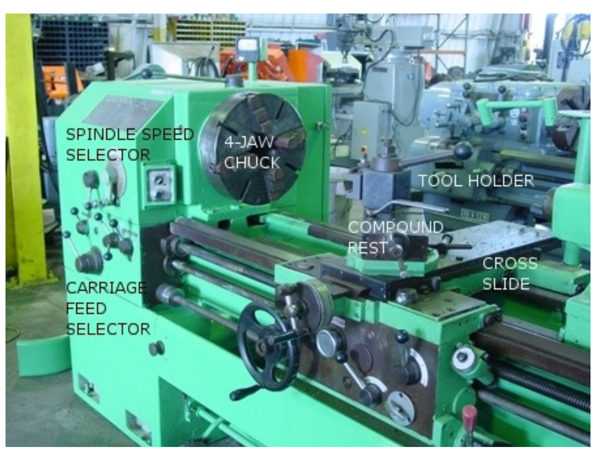

Figure 1.3.2: Engine Lathe

1.3.1 Components of a lathe:

Chuck: this is an oversize version of the chucks used in drill presses and mills; chucks are equipped with 3 or 4 jaws.

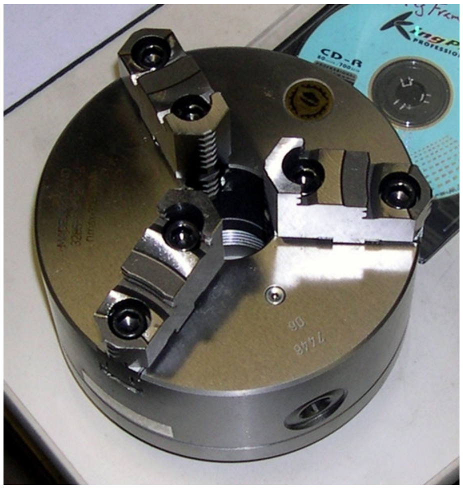

Figure 1.3.3: Three-Jaw Chuck

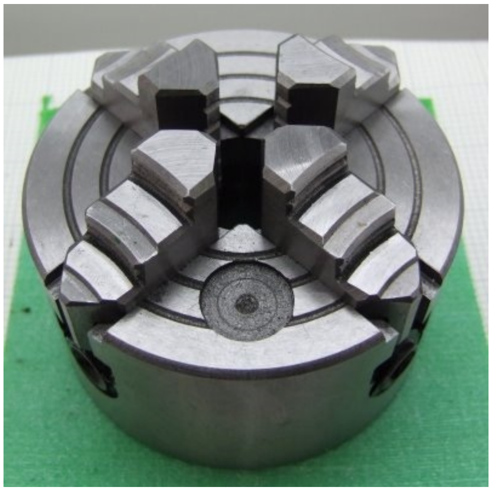

Figure 1.3.4: Four-Jaw Chuck

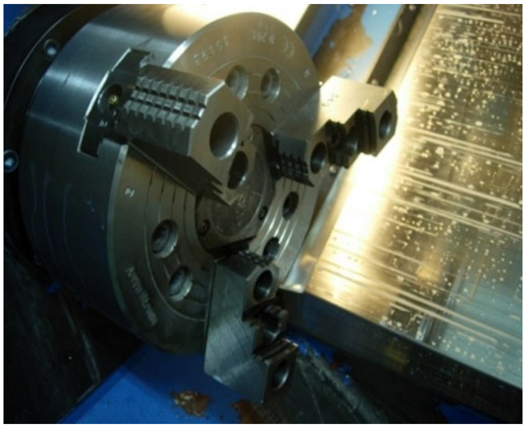

Figure 1.3.5: Three-Jaw Chuck with Jaws Reversed

The 3-jaw chuck moves all 3 jaws simultaneously, thereby automatically centering the work-piece; the jaws on a 4-Jaw Chuck are moved individually, which makes centering much more difficult, but also allows for so-called “off-centre” set-ups.

Figure 1.3.5 shows a 3-jaw chuck with the jaws flipped 180 degrees to allow larger diameter work-pieces to be mounted.

Spindle: the spindle in a lathe “only” rotates, it does not move in an axial direction; the chuck is again mounted on the spindle.

Head stock: it houses the spindle as well as the gears and pulleys needed to set the speeds and feeds.

Carriage: moves left and right, parallel to the spindle axis.

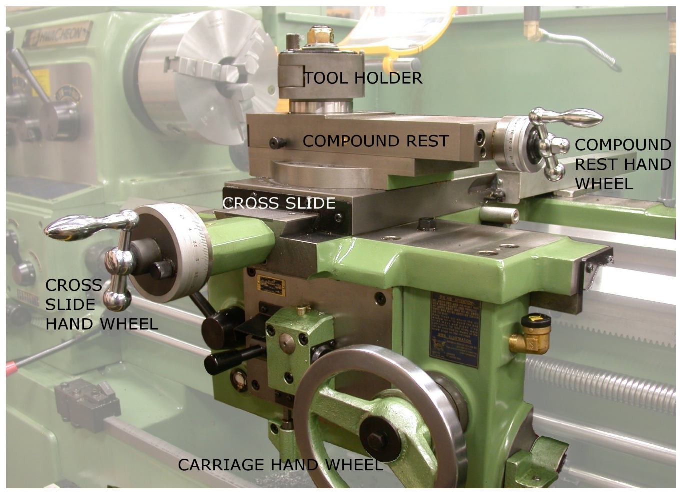

Figure 1.3.6: Carriage, Cross Slide, Compound Rest and Tool Holder

Cross slide: is mounted on the carriage and can be moved in a direction perpendicular to that of the carriage (moves “in and out”).

Compound rest: sits on top of the cross slide on a rotating base and can be moved forwards or backwards.

Tool holder: is bolted to the top of the compound rest and is used to mount the cutting tool.

Carriage, cross-slide, compound rest and tool holder are the components which, together, position and move the tool to turn the desired shape.

Tail stock: is used to support the work piece or hold drilling or boring tools.

1.3.2 Lathe tools:

There are 2 major types of lathe cutting tools: those made from HSS and those using carbide inserts.

HSS tools are normally made from HSS blanks and are ground to the desired shape by the operator. Figure 1.3.7 shows the geometry of such a cutting tool, Figure 1.3.8 shows the HSS blanks which are used to make those tools.

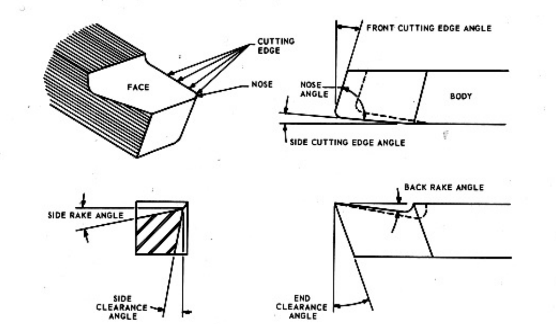

Figure 1.3.7: HSS Tool Geometry


Figure 1.3.8: HSS Tool Blanks

The magnitude of the tool angles depends on the material of the work-piece and the type of cut to be performed; this information can be found from tables (Machinery's Handbook).

Carbide inserts are mounted on tool bars, as shown in Figure 1.3.9. Similar to HSS tools, inserts are produced with different nose geometry.


Figure 1.3.9: Carbide Inserts and Tool Bars

Both types of tools are securely mounted in the tool holder.

Whereas HSS tools can be sharpened or re-ground, carbide inserts are discarded after all cutting corners have been dulled.

Other, less common tools, mounted on the tool holder are knurling tools, parting tools and thread cutting tools.

Figure 1.3.10: Knurling Tools

Figure 1.3.11: Parting Tool

Thread Cutting tools look very similar to conventional cutting tools, but the tool geometry and angles are different. The knurling tool produces a diamond-shaped pattern on the surface of a turned part. Parting tools are used to cut off the turned part from the remaining piece of stock.

The tools discussed so far all are mounted on the tool holder (carriage) and can travel longitudinally as well as transversely or a combination of both.

But tools can also be mounted in the tail stock; those tools are drills, reamers, boring bars etc. They are used to produce central holes in the work-piece.

1.3.3 Lathe cutting speeds in "surface feet per minute" [sfm]:

"Surface feet per minute" describes the linear (tangential) relative velocity of the part surface with respect to the tool.

WORK MATERIAL		Bhn	HSS [sfm]	CARBIDE [sfm]
Plain Carbon Steel: AISI 1010 to 1025	HR	100 - 125	140	500
	HR	125 - 175	120	400
	HR, CD	175 - 225	100	350
	CD	225 - 275	70	300
AISI B1111, AISI B1112, AISI B1113 etc.	HR	100 - 150	160	500
	CD	150 - 200	180	600
AISI 1108, 1115, 1118, 1120, 1126	HR	100 - 150	140	450
	CD	150 - 200	150	500
AISI 1132,1137,1140, 1145, 1151	HR	175 - 225	130	500
	Q&T	275 - 325	90	250
	Q&T	325 - 375	50	175
	Q&T	375 - 425	30	140
AISI 1027, 1029, 1030, 1032, 1035, 1037,	HR, CD	125 - 175	120	400
1040, 1043, 1045,1047, 1050	HR, CD	175 - 225	100	350
	CD, Q&T	225 - 275	70	300
	Q&T	275 - 325	60	240
	Q&T	325 - 375	50	200
	Q&T	375 - 425	40	175
AISI 1055, 1060, 1070, 1074, 1080	HR, CD	125 - 175	100	375
1085, 1090, 1095	HR, CD	175 - 225	90	325
	CD, Q&T	225 - 275	65	275
	Q&T	275 - 325	55	225
	Q&T	325 - 375	45	180
	Q&T	375 - 425	30	150
AISI 3140, 4140, 4150, 8640	HR, CD	175 - 200	125	450
	HR, CD	200 - 250	100	400
	Q&T	250 - 300	70	325
	Q&T	300 - 375	60	225
	Q&T	375 - 425	40	150
Alloy Steels: AISI1320, 2317, 2512, 2517, 3115,	HR, CD	150 - 175	110	400
3120, 3125, 3310, 3316, 4012, 4017, 4023, 4028,	HR, CD	175 - 220	80	350
4320, 4615, 4620, 4720, 4815, 4820, 5015, 5020,	CD, Q&T	220 - 275	70	300
5024, 5120, 6118, 6120, 6317, 6325, 6415, 8115,	Q&T	275 - 325	60	250
8615, 8620, 8625, 8720, 8822, 9310, 9315	Q&T	325 - 375	50	200
	Q&T	375 - 425	40	175
Aluminum			300	800
Cast Iron			140	325
Stainless Steel, Annealed Tool Steel			50	200
Yellow Brass			200	500
Plastics			500	1000

Table 1.3.1: Turning Speeds [sfm]
HR.....Hot Rolled, CD......Cold Drawn, Q&T.....Quenched and Tempered

As before, based on the cutting speeds from Table 1.3.1 and knowing the initial work-piece diameter, we can calculate the spindle rpm, using the modified “formula”:

$$\frac{(Cutting\ Speed\ [sfm])\ x\ 4}{(Diameter\ of\ Work\ piece\ [in])} \cong Spindle\ Speed\ [rpm]$$

Note that in this case we need to use the initial diameter of the work-piece; if you are machining significantly different diameters on one piece, then it may be necessary to set different spindle speeds for each diameter.

1.3.4 Lathe Feeds:

Material	**HSS [in/rev]**		**Carbide [in/rev]**	
	Roughing	Finishing	Roughing	Finishing
Low Carbon Steel	.010 - .020	.002 - .008	.008 - .035	.006 - .010
Medium Carbon Steel	.008 - .018	.002 - .008	.008 - .030	.006 - .010
High Carbon Steel	.008 - .015	.002 - .008	.008 - .030	.006 - .010
Cast Iron	.010 - .025	.003 - .010	.010 - .040	.008 - .012
Bronze, Brass	.015 - .025	.003 - .010	.010 - .040	.008 - .012
Aluminum	.015 - .030	.003 - .012	.015 - .045	.008 - .012

Table 1.3.2: Turning Feeds [in/rev]

Using the “formula”

$$(Spindle\ RPM)\ x\ (Feed\ [in/rev]) = Feed\ Rate[in/\min]$$

we can now set the lathe to the required spindle speed [rpm] and tool feed [in/min].

Example: *A shaft, finished diameter = 1.250", is to be fabricated from a 1.50" diameter round bar made of AISI 1020 low carbon steel, Bhn 180*. *The first (roughing) cut will be 0.010" deep*: **this will reduce the part diameter from 1.50" to 1.30".**
The finishing cut will be 0.0025" deep: **this will reduce the part diameter to 1.25".**

On most lathes the radial feed of the tool indicates the true distance the tool has been moved inward (which reduces the part diameter by twice that amount); some lathes indicate the effective change in part diameter. Therefore: check before moving your tool.

Find the spindle speed and carriage feed settings. Both cuts will be made with a carbide tipped tool.

From Table 1.3.1: for AISI 1020, Bhn 180, the cutting speed is 350 [sfm] and using the formula

$$\frac{(Cutting\ Speed\ [sfm])\ x\ 4}{(Diameter\ of\ Work\ piece\ [in])} \cong Spindle\ Speed\ [rpm]$$

we can calculate the spindle rpm:

$$\frac{350\ [sfm]\ x\ 4}{1.5''} = 933.3\ [rpm]$$ *roughing cut rpm and*

$$\frac{350\ [sfm]\ x\ 4}{1.25''} = 1120\ [rpm]$$ *finishing cut rpm.*

The available spindle speeds will probably be 900 [rpm] and 1100 [rpm] (depending on availability).

From Table 1.3.2: for low carbon steel and a carbide tool the roughing feed is .008 - .035 [in/rev], the finishing feed is .006 - .010 [in/rev]; using the "formula"

$$(Spindle\ RPM)\ x\ (Feed\ [in/rev])\ = Feed\ Rate[in/\min]$$

we can calculate the feed rates [in/min]:

$$900\ [rpm]\, x\, .020\ [in/rev] = 18[in/\min] \quad roughing\ \ feed\ \ rate$$

$$1100\ [rpm]\, x\, .008\ [in/rev] = 8.8\ [in/\min] \quad finishing\ \ feed\ \ rate$$

So that the final settings for turning the shaft will be:

Roughing spindle speed:	*900 [rpm]*
Finishing spindle speed:	*1100 [rpm]*
Roughing feed rate:	*18 [in/min]*
Finishing feed rate:	*8 [in/min]*