1. MACHINING OPERATIONS:

1.1 DRILL PRESS:

The main function of a drill press is the drilling of holes up to a diameter of approximately 20 mm (or ~3/4 inches), depending on the material the holes are being drilled into.

Figure 1.1.1 shows a small, floor-mounted, manual drill press.

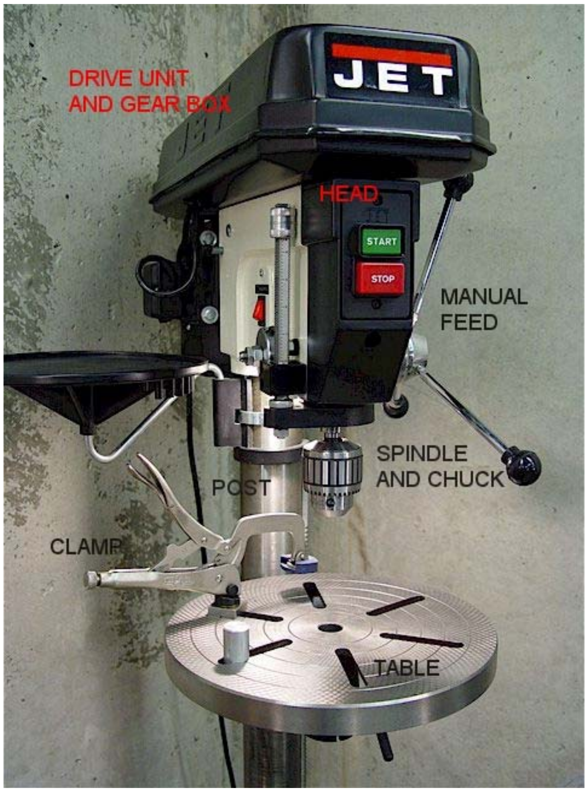

Figure 1.1.1: Manual Drill Press

1.1.1 Components of a drill press:

Drive unit and gear box: this gear box allows the selection of the correct drill (rotational) speed; drill speeds are a function of the type and diameter of drill and the material being drilled into.

Chuck: is a device which holds standard tool bits. By inserting the chuck key into one of the radial holes in the chuck, the jaws of the chuck can be opened or closed.

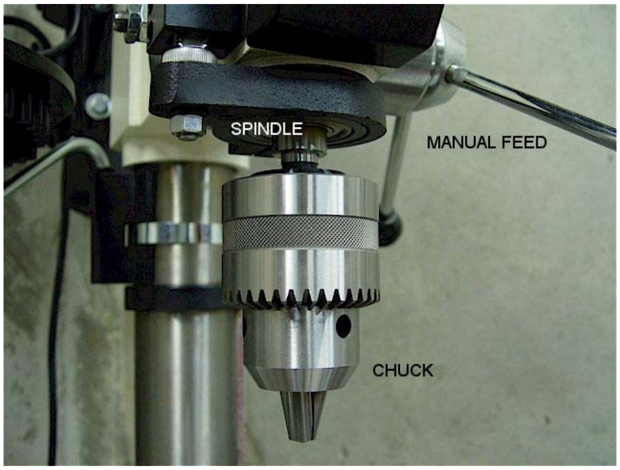

Figure 1.1.2: Standard Chuck

Figure 1.1.3: Various Chucks

Spindle: it holds the chuck or the tapered end of the drilling tool and can be moved vertically by turning the vertical feed wheel/arms.

Head: this is the housing that contains the spindle, sleeve, bearings etc.

Manual vertical feed: moves the spindle (drill) vertically up and down by manually rotating the arms of the vertical feed.

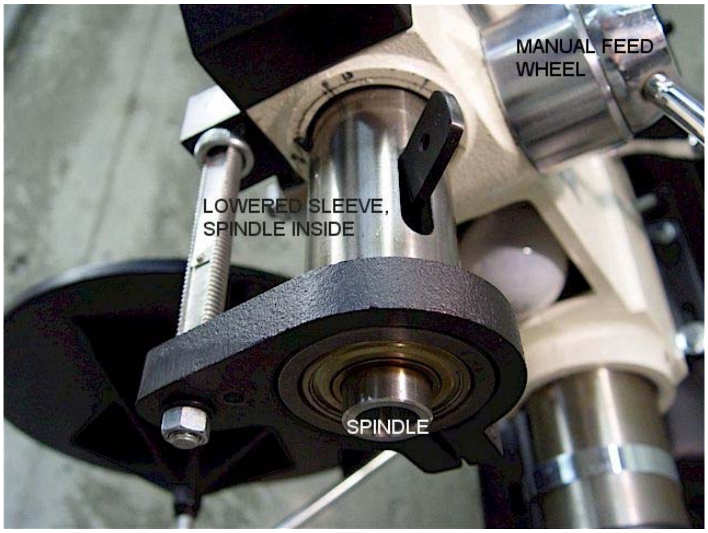

Figure 1.1.4: Sleeve and Spindle

Vise or Clamp: devices used to securely hold the workpiece to the table.

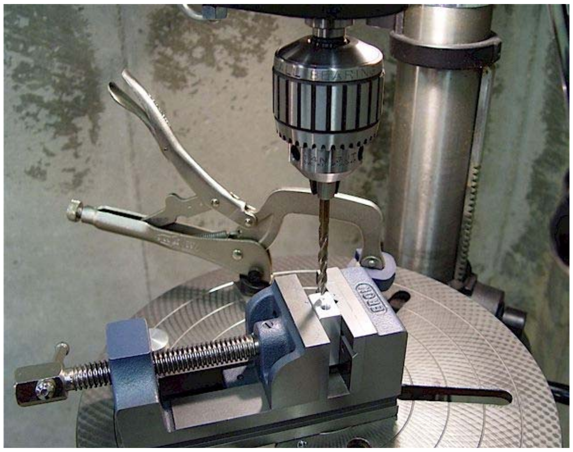
Figure 1.1.5: Machine Vise


Figure 1.1.6: Hold-down Clamp

Table: a flat surface that is perpendicular to the spindle axis and supports the workpiece or the device holding the workpiece.

1.1.2 Drilling tools:

The most commonly performed operation on a drill press is drilling a hole. The actual tools used for this operation are called drill bits. Depending on the type of material and the type of hole required, different types of drill its are available.

The most common type of drill bit: the straight-shank twist drill

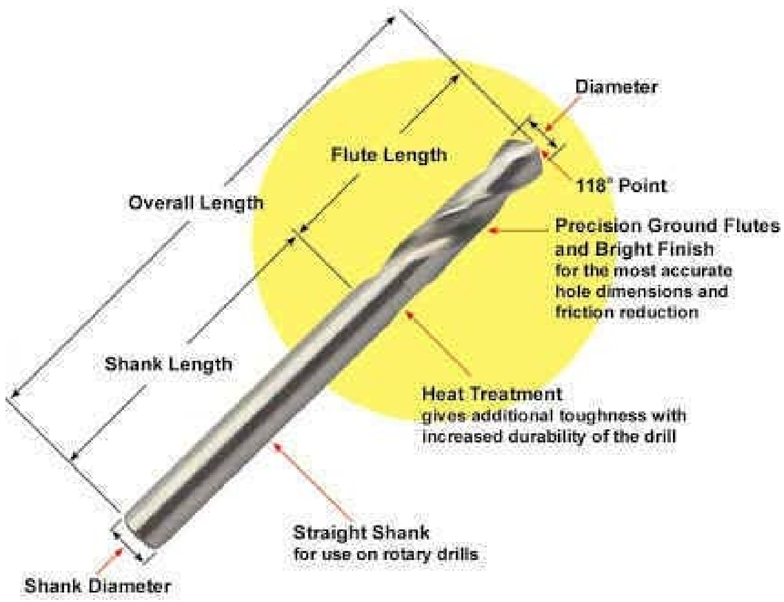

Figure 1.1.7: Twist Drill

For this type of drill the drill diameters can be specified in fractional inches, millimetres, letters or numbers.

Fractional sizes:	from 1/64" to 1" (in increments of 1/64" for the medium to larger sizes)
Millimetres:	from 0.2 mm to 25 mm in increments of 0.2 mm (specialty sets in increments of 0.1 mm)
Number sizes:	those are inch-size drills, starting with #80 (= 0.104") to #1 (= 0.228")
Letter sizes:	this is in effect a continuation of the number sizes, except they are identified by letters from A to Z (where A = 0.2340" and Z = 0.4130").

A complete chart of all these drill bit sizes can be found at:

http://www.carbidedepot.com/formulas-drillsize.htm or similar pages.

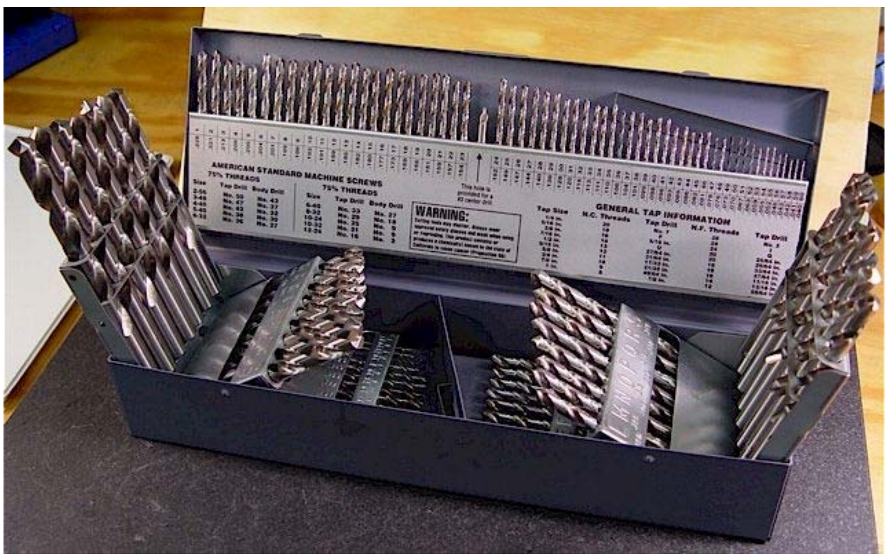

Figure 1.1.8: Set of Inch Bits: 1/16" - ½", #60 - #1 and A - Z

Note that these larger drill bits, shown in Figure 1.1.9, have shanks which are ½"; this way they can be mounted in standard chucks.


Figure 1.1.9: ½" to 1" Bits with ½" Shank

Larger size drill bits:

Larger size and longer drill bits are available in such a form that they can be mounted directly into the spindle of the drill press (after removing the chuck).

The tapered shank centres the bit perfectly in the spindle, whereas the tang, the flat piece at the end of the taper, fits into a mating slot in the spindle and transmits torque.

Figure 1.1.10: Tapered Shank with Tang Drill Bits ½" to 3/4"

Twist drill bits can be made from different materials:

High-Speed-Steel (HSS):	most common type for working with materials no harder than mild steel (black or silver in appearance).
Cobalt:	made from 3% to 8% cobalt HSS; they can be used for high tensile strength metals (inconel, titanium and stainless steel)
TiN-coated:	this is a standard HSS bit with a Titanium Nitride coating which reduces friction and extends the life of the bit. Used for the same materials as HSS.
Carbide:	capable of drilling into most materials; much longer life than HSS.

1.1.3 Drilling speeds:

Drilling speeds refer to the revolutions per minute [rpm] of the spindle. Depending on the material being drilled into and the type and size of drill bit used, different speeds must be used. Cutting speeds are typically given in “sfm” (surface feet per minute) and refer to the distance travelled by a point on the cutting edge of the drill bit in one minute.

From these speeds one can then calculate the required spindle speed [rpm].

$$\frac{(Cutting\ \ Speed\,[sfm])\ x\,12}{(Tool\ \ Diameter\,[in])\ x\ \pi} = Spindle\,Speed\,[rpm]$$

which can be simplified to

$$\frac{(Cutting\ \ Speed\,[sfm])\ x\,4}{(Tool\ \ Diameter\,[in])} \cong Spindle\,Speed\,[rpm]$$

Should you have to use metric cutting speed tables, then use the following “formula”:

$$\frac{(Cutting\,Speed\,[m/\min])\ x\,320}{Tool\ \ \ Diameter\,[mm]} = Spindle\,Speed\,[rpm]$$

In general: the softer the material and the harder the tool (from HSS to Carbide) the higher the cutting speed.

Cutting Speeds for most materials can be found in publications such as the “Machinery’s Handbook” or tool manufacturers’ specifications.

MATERIAL	Hardness [Bhn]	Cutting Speed [sfm]	Cutting Speed [m/min]	MATERIAL	Hardness [Bhn]	Cutting Speed [sfm]	Cutting Speed [m/min]
Plain Carbon Steels AISI-1019, 1020 to 1090	120–150 150–170 170–190 190–220 220–280 280–350 350–425	80–120 70–90 60–80 50–70 40–50 30–40 15–30	25 - 37 21 - 27 18 - 25 15 - 21 12 - 15 9 - 12 5 - 9	Tool Steels Water Hardening Cold Work Shock Resisting Mold High-Speed Steel	 150–250 200–250 175–225 100–150 150–200 200–250 250-275	 70–80 20–40 40–50 60–70 50–60 30–40 15–30	 21 - 25 6 - 12 12 - 15 18 - 21 15 - 18 9 - 12 5 - 9
Alloy Steels AISI-1320, 2317, 2515, 3120, 3316, 4012, 4020, 4120, 4128, 4320, 4620, 4720, 4820, 5020, 5120, 6120, 6325, 6415, 8620, 8720, 9315	125–175 175–225 225–275 275–325 325–375 375–425	60–80 50–70 45–60 35–55 30–40 15–30	18 - 25 15 - 21 14 - 18 11 - 16 9 - 12 5 - 9	Gray Cast-Iron	 110–140 150–190 190–220 220–260 260–320	 90–140 80–100 60–80 50–70 30–40	 27 - 42 24 - 30 18 - 25 15 - 21 9 - 12
Alloy Steels AISI-1330, 1340, 2330, 2340, 3130, 3140, 3150, 4030, 4063, 4130, 4140, 4150, 4340, 4640, 5130, 5140, 5160, 52100, 6150, 6180, 6240, 6290, 6340, 6380, 8640, 8660, 8740, 9260, 9445, 9840, 9850	175–225 225–275 275–325 325–375 375–425	50–70 40–60 30–50 25–40 15–30	15 - 21 12 - 18 9 - 15 7 - 12 5 - 9	Malleable Iron Ferritic Pearlitic	 110–160 160–200 200–240 240–280	 120–140 90–110 60–90 50–60	 36 - 42 27 - 34 18 - 27 15 - 18
Stainless Steels Standard Grades Austenitic Annealed Cold-Drawn Ferritic Martensitic Annealed	135–185 225–275 135–185	40–50 30–40 50–60	12 - 15 8 - 12 15 - 18	Aluminum Alloys		 200–300 150–250 150–300 140–300	 60 - 90 45 - 75 45 - 90 42 - 90
				Brass & Bronze (Ordinary)		150–300	45 - 90
				Bronze (High Strength)		30–100	8 - 30

Table 1.1.1: HSS Cutting Speeds in [sfm] and [m/min]

Table 1.1.1 shows recommended cutting speeds for HSS tools; coated HSS tools use the same cutting speeds.

Carbide (or carbide tipped) tools use cutting speeds which are approximately 3 times higher than those for HSS; but since different grades of carbide are used for tools, check with the manufacturer (if you are trying to optimize the cutting speed).

Table 1.1.2 shows recommended feed rates (the rate at which the drill bit moves vertically) in [in/rev] or multiply by 25 to get [mm/rev]. To arrive at the actual linear speed with which the drill should move into the work-piece, multiply the feed rate by the spindle speed [rpm].

The following feed rates are again for HSS tools.

Drill Diameter [in]	Feed Rate [in/rev]
1/16 - 1/8	.001 - .003
1/8 - 1/4	.002 - .006
1/4 - ½	.004 - .010
½ - 1	.007 - .015
over 1	.015 - .025

Table 1.1.2: HSS Feed Rates [in/rev]

Example: *You have to drill a 0.250" diameter through hole into plain, 1" thick, carbon steel, Bhn 300, using a HSS tool. Find the spindle speed and the approximate vertical feed rate for the operation.*

From Table 1.1.1:
recommended cutting speed = 35 [sfm]

With:

$$\frac{(Cutting\ \ Speed\,[sfm])\ x\ 4}{(Tool\ \ Diameter\,[in])} \cong Spindle\,Speed\,[rpm]$$

$$\frac{35[sfm]\,x\,4}{0.250''} = 560[rpm]$$

Select the closest available spindle speed.

Example continued:

From Table 1.1.2:
Recommended feed rate = from 0.004 to 0.006 [in/rev]

For a selected spindle speed of 500 [rpm] this would produce a vertical displacement of ~3 inches in one minute. This means for our 1" thick piece it should, theoretically, take about 20 seconds to produce a through hole.
It will actually take longer because of a process called pecking: it is recommended that, when drilling a hole, the drill bit is only moved approximately 0.2" into the work-piece and is then "pulled back" (moved back outside the hole) so that chips don't build up inside the flutes of the tool; it is then cycled back into the hole and the process is repeated until the hole has been drilled through.

1.1.3 Other drill press tools:

The other types of tools, other than the common drill bit, that can be used on a drill press:

- -fly cutters (boring bars)
- -reamers
- -counter-boring tools
- -counter-sinking tools

Fly cutters and boring bars:

These tools are typically used when over-sized or odd-sized holes have to be fabricated.

Figure 1.1.11: Fly Cutter

Figure 1.1.12: Boring Bars

In both cases the tip of the tool can be adjusted radially to produce the desired hole diameter. Boring Bars are more effectively used for turning operations.

Reamers:

Reamers are tools which are capable of producing holes to much tighter tolerances, but they can only be used as a “refinement” tool: first drill a hole using a standard drill bit, then replace the drill bit with a, slightly larger, reamer (adjust the cutting speed and the feed rate) and “open up” the hole to the desired size.
Reamer sizes are typically ± 0.0010" from a standard size. For example: for a standard size of 1/4" (= 0.250") reamers would be available in sizes of 0.2510" and 0.2490" (significant figures intentional).
Cutting speeds for reaming operations should be from 1/3 to 2/3 of the drilling speed; feed rates should be very low.

Figure 1.1.13: Reamers

Counter-bores:


Figure 1.1.14: Counter-bored Hole


Figure 1.1.15: Counter-bores

Figure 1.1.14 shows a counter-bored hole: the larger diameter part of the hole allows the head of a bolt (or similar fastener) to be “hidden” below the top surface of the part. Counter-boring tools are identified by the hole diameter: for example to produce a counter-bore for a 6 mm diameter hole, you would select a M6 counter-bore (the diameter of the counter-bore part is standardized).

To fabricate a counter-bored hole: drill a hole of the required size first and then replace the drill bit with the appropriate counter-bore bit; using the depth gauge on the drill press, and after adjusting the cutting speed, counter-bore to the desired depth.

Counter-sinks:


Figure 1.1.16: Counter-sunk Holes


Figure 1.1.17: Counter-sink

Counter-sinking is a conical enlargement of a hole to a certain depth, so that the conical head of a screw or fastener can be set flush to or slightly below the surface of a part. Counter-sinks come in standardized cone angles.
Similar to reaming and counter-boring, a conventional hole has to be drilled first before counter-sinking to a certain depth or a certain diameter can be performed.