Drilling

Quick Summary:

Some approximate **starting** advice is to set the spindle speed between 700-1000 rpms for steel, above 2000 for aluminium, and adjust from there if you get discoloured chips or heavy drill bit wear. In most cases the drill press will not be able to supply enough power or speed to follow the below recommendations. Take for instance a 9.5mm Drill bit drilling mild steel at the recommended speed of 1019r/min. Going slower usually doesn't hurt, and will prolong tool life.

Drill press spindle speeds depend on lots of things:

1. Composition, hardness, and thermal conductivity *(k)* of material
2. Depth of hole and hole size
3. Efficiency of cutting fluid
4. Stiffness and condition of drilling machine
5. Stiffness of work piece, fixture, and tooling (shorter is better)
6. Quality of holes desired
7. Life of tool before regrind or replacement

In your workshop

- So, for the hobbyists shop, where longer tool life is probably more important than machining time, and where pushing the speed limit may ruin a valuable prototype, reasonable advice might be to **start off at about 75% of the recommended drilling speeds.** The "First Guesses" below are already a little slower.

- You'll typically see large ranges of recommended speeds for various materials, and some discrepancy between different sources. This is partly due to the large influence the material hardness has on how fast it can be drilled (harder --> slower). Even if the material and its hardness were precisely known, the large number of other factors would require some experimentation. **If the chips are smoking, turning brown or the outer edge of the drill bit is chipping, go slower or add some cutting oil / coolant.**

- **In general, a slower-than-recommended spindle speed won't hurt anything** except in the case of extremely small drill bits; say smaller than 1.5mm. With small bits, it's hard to feel resistance from the metal, and therefore, very easy to push down faster than they can remove metal. Using recommended RPMs (spindle rotation speed) mitigates this risk. A tip for drilling extremely small holes is to drill down to the depth, stop, take the drill bit out, and repeat. This ensures that too much metal isn't chewed off too quickly.
-
- **Feed Rate:** We are all drilling by hand. Listen when drilling, it will tell you when it is taking away any materials. Push hard enough to create a continuous chip (note some materials just won't form one--like cast iron), but not so hard that the chips are turning brown or the bit itself is chipping. When the bit is about to break through, it will go silent!!!! Take the pressure off the drill. This will reduce the likelihood of it grabbing and tearing the metal.

- **Surface Feet per Minute (SFM):** Speed recommendations are usually given in SFM, which is the speed a cutter can be pushed in a straight line. On drill bits, the fastest cutting rate is at the circumference, and its rate of travel is equal to the RPM (Revolutions Per Minute) of the bit times the circumference (pi*Diameter). So, the translation between SFM and the RPM speed of a drill bit is:

The SFM will be given by the manufacturer of the drill

Material	Speed Range (SFM)	First Guess (SFM)	RPM recommendations based on First Guess Speeds for various drill bit diameters					
			3.175mm	6.35mm	7.93mm	9.5mm	11mm	12.5mm
low carbon steel,	60-100	100	3056	1528	1222	1019	873	764
high carbon / alloy	55-85	50	1528	764	611	509	437	382
tool steel	45-60	50	1528	764	611	509	437	382
cast Iron	50-125	70	2139	1070	856	713	611	535
aluminium and alloys	200-300	250	7639	3820	3056	2546	2183	1910
brass / bronze high strength bronze may require 70 or less	150-300	200	6112	3056	2445	2037	1746	1528
wood	300-400	300	9167	4584	3667	3056	2619	2292

Some considerations for using the above table:

- First, note that the speed recommendations for small bits in aluminium are ridiculously high compared to the max speed of around 3000 RPMs on bench-top drill presses. These numbers are more just for reference, its fine to go slower, just don't push too hard on the small bits.
- "First guesses" are based on more typical materials and hardness
- If the material has been hardened, the recommended speed will be substantially lower. If the hardness is above 32 Rockwell, starting at 20-30 SFM isn't a bad idea.
- Note that these values are recommended for HSS (High Speed Steel) drill bits, not carbon steel ones. HSS gets its name because it is able to maintain a reasonably hard cutting edge even while it is red hot. If you're using carbon steel drill bits (unlikely), cut the recommendations in half.
- Adding cooling / cutting fluid may allow speed increases, too, and should be used in any case on steels. Cutting fluid will almost always increase the quality of the cut.