EML2322L – MAE Design and Manufacturing Laboratory

Drilling Speeds and Feeds

The speed of a drill is measured in terms of the rate at which the outside or periphery of the tool moves in relation to the work being drilled. The common unit and term for this velocity is *surface feet per minute,* abbreviated sfm. Every tool manufacturer has a recommended table of sfm values for their tools. General sfm guidelines are found in resources such as the Machinery Handbook (see Table 1 in this document for a condensed version).

The peripheral and rotational velocities of the tool are related as shown in the following equation:

$$V = \pi \times D \times N \qquad (Eq.\ 1)$$

where

V is the recommended peripheral velocity for the tool being used
D is the diameter of the tool
N is the rotational velocity of the tool

Since the peripheral velocity is commonly expressed in units of feet/minute and tool diameter is typically measured in units of inches, Equation 1 can be solved for the spindle or tool velocity, *N* in the following manner:

$$N\ [rpm] = 12\ [in/ft] \times V\ [sfm] \,/\, (\pi \times D\ [in/rev]) \qquad (Eq.\ 2)$$

Equation 2 will provide a guideline as to the **maximum speed** when drilling standard materials. The **optimum speed** for a particular setup is affected by many factors, including the following:

1. composition, hardness, and thermal conductivity *(k)* of material
2. depth of hole
3. efficiency of cutting fluid
4. stiffness and condition of drilling machine
5. stiffness of workpiece, fixture, and tooling (shorter is better)
6. quality of holes desired
7. life of tool before regrind or replacement

Table 2 contains recommended feed rates for various drill diameters. For each diameter range there is a corresponding feed range. **Use the smaller values for stronger/harder materials and the larger values for weaker/softer materials.** To calculate the feed rate, use the following formula:

$$f = N \times f_r \qquad \text{(Eq. 3)}$$

where

f = calculated linear feed rate of the drill [in/min]
N = spindle speed [rpm]
f_r = feed per revolution of the drill [in/rev]

In addition, the following rules of thumb should be observed when applicable:

- Always **start with a slower speed (~50%) and build up to the maximum** after trials indicate the job can run faster.

- **Overloading the drill bit by feeding too quickly in Z will result in an excessive chipload on each drill lip, causing the cutting edges to fracture (chip).**

- **Peck drilling**, or the practice of drilling a short distance, then withdrawing the drill, **will reduce the tendency of chips to collect in the bottom of the hole.** The deeper the hole, the more frequent the drill must be retracted (or pecked) to be effective.

- The deeper the hole, the greater the tendency is for chips to pack and clog the flutes of the drill. This increases the amount of heat generated and prevents the cutting lubricant from conducting heat away from the drill point (where all the cutting occurs). Excessive buildup of heat at the drill point results in premature failure. Therefore, a reduction in speed and feed to reduce the amount of heat is required in deep-hole applications where coolant cannot be effectively applied. Consequently, **feeds and speeds should be reduced up to 50% when drilling holes deeper than 3 drill diameters.**

Table 1: Recommended HSS Speeds for Common Materials

Material	Recommended Speed, *V* [surface ft/min]
Aluminum and its alloys	250
Brass	250
Bronze (high tensile)	100
Cast Iron (soft)	100
Cast Iron (medium hard)	80
Cast Iron (hard chilled)	20
Hastelloy	20
Inconel	25
Magnesium and its alloys	300
Monel	25
High nickel steel	50
Mild steel (.2-.3 C)	100
Steel (.4-.5 C)	60
Tool steel	40
Forgings	40
Steel alloys (300-400 Brinell)	30
Heat Treated Steels	
35-40 Rockwell C	20
40-45 Rockwell C	20
45-50 Rockwell C	15
50-55 Rockwell C	15
stainless steel (free machining)	40
stainless steel (work hardening)	20
Titanium alloys	20

* Multiply surface speeds in table by 2.5 for carbide cutting tools. *

Table 2: Recommended Average Feed Rates for 2 Flute HSS Drills

Drill Diameter [in.]	**Recommended Feed, f_r [in./rev]**
under 1/8″	up to 0.002
1/8″ to 1/4″	0.002 to 0.004
1/4″ to 1/2″	0.004 to 0.008
1/2″ to 1″	0.008 to 0.012
1″ and over	0.012 to 0.020

Final Notes:

- Remember that the speed and feed calculated using the manufacturer's empirical data (i.e. Tables 1 & 2) are the ***optimum*** parameters. In other words, these are the ***maximum*** speed and feed rate that could be used under ***perfect conditions*. To promote their products, this published data is usually optimistic** (i.e. the speeds and feeds are typically on the high side). Manufacturers will generate the data using the stiffest machines and workpiece setups available, very high pressure coolant (1000+ psi) or highly effective (and expensive) cutting oil, etc.

- **Running a tool too slow will only decrease productivity; however, running a tool too fast with regard to speed or feed rate will result in accelerated tool wear or outright failure.** So always err on the side of running too slow in a laboratory environment. (A production environment involves an entirely different set of objectives.)

Example: Calculate the speed and feed for a ¼″ HSS drill bit in soft cast iron on a manual milling machine in the lab.

First, lookup the recommended surface speed (peripheral velocity) in Table 1:

$V \approx 100$ ft/min

Next, calculate the spindle speed from Equation 2:

$N\ [rpm] = 12 \times V / (\pi \times D)$
$= 12$ in/ft $\times$ 100 ft/min / ($\pi \times 0.25$ in/rev)
$\approx$ **1500 rpm (ans)**

Now calculate the feed rate used for plunging in the Z axis:

From Table 2, lookup the recommended feed per revolution for a 1/4″ HSS drill bit:

$f_r \approx 0.004$ in/rev

Finally, calculate the plunge feed rate using Equation 3:

$f\ [in/min] = N\ [rpm] \times f_r\ [in/rev]$
= 1500 rev/min $\times$ 0.004 in/rev
= **6.0 in/min (ans)**

Note that these speed and feed values are guidelines assuming adequate (flooded) lubrication, workpiece stiffness and drill depth less than 3 drill diameters (0.75″). When applying oil manually (as in the lab), scale the feed and speed back to 60%, so ***N* = 900 rpm and *f* = 3.6 in/min (final ans)**.

Milling Speeds and Feeds

Like a drill bit, the speed of a milling cutter is measured in terms of the rate at which the outside, or periphery, of the tool moves in relation to the work being milled. General sfm guidelines are found in resources such as the Machinery Handbook (refer again to Table 1 in this document).

The peripheral and rotational velocities of the tool are related by the same equation:

$$V = \pi \times D \times N \qquad \text{(Eq. 1)}$$

where

- V is the recommended peripheral velocity for the tool being used
- D is the diameter of the tool
- N is the rotational velocity of the tool

Since the peripheral velocity is commonly expressed in units of feet/min and tool diameter is typically measured in units of inches, Equation 1 can be solved for the spindle or tool velocity, N in the following manner:

$$N\ [rpm] = 12\ [in/ft] \times V\ [sfm]\ /\ (\pi \times D\ [in/rev]) \qquad \text{(Eq. 2)}$$

Equation 2 will provide a guideline as to the **maximum speed** when milling standard materials. The **optimum speed** for a particular setup is affected by many factors, including the following:

1. composition, hardness, and thermal conductivity *(k)* of material
2. depth of cut (roughing or finishing)
3. efficiency of cutting fluid
4. type, condition and stiffness of milling machine
5. stiffness of workpiece, fixture, and tooling (shorter is better)
6. quality of finish desired
7. life of tool before regrind or replacement

In addition, the tool manufacturer will provide information regarding the feed per tooth (or chipload) for their specific cutters. See Table 3 for an example. To calculate the feed rate to use once the rpm has been computed, use the following formula:

$$f = N \times f_t \times m \qquad \textit{(Eq. 4)}$$

where

f = linear feed rate of the endmill / cutter [in/min]
N = spindle speed [rpm]
f_t = feed per tooth of the endmill / cutter [in/tooth]
m = number of teeth on endmill / cutter [integer]

Once again, the initial choice of speed and feed will be at the lower value of the ranges listed and will ultimately depend upon the following variables:

FEEDS	
Use lower feed range for:	Use higher feed range for:
Fine tooth cutters Fragile and small cutters Light and finishing cuts Deep slots Hard-to-machine materials Flexible workpiece or setup	Coarse tooth cutters Rugged cutters Heavy roughing cuts Abrasive materials Easy-to-machine materials Rigid setups
SPEEDS	
Use lower speed range for:	Use higher speed range for:
Heavy cuts Hard materials Tough materials Abrasive materials Maximum cutter life Minimum tool wear Flexible workpiece or setup	Light cuts Softer materials Non-metallics Maximum production rates Better finishes Hand feed operations Higher tool rake angles

Table 3: Recommended Feeds for HSS Milling

Material	Recommended Feed, f_t [in./tooth]
Aluminum	0.004 to 0.012
Magnesium	0.004 to 0.012
Brass (soft)	0.004 to 0.012
Brass (hard)	0.002 to 0.010
Bronze (soft)	0.004 to 0.012
Bronze (hard)	0.002 to 0.010
Copper	0.004 to 0.012
Steel	
100 BRN	0.004 to 0.006
200 BRN	0.004 to 0.005
300 BRN	0.003 to 0.004
400 BRN	0.002 to 0.003
500 BRN	0.001 to 0.002
Stainless (free machining)	0.002 to 0.005
Stainless (hard)	0.002 to 0.004
Titanium (soft)	0.002 to 0.005
Titanium (hard)	0.001 to 0.003

* Table values are typical for cutters ranging from ½″ to 1-1/2″ in size. *

Example: Calculate the speed and feed for a 1″ diameter, 4 flute HSS endmill in aluminum using a manual milling machine in lab.

First, lookup the recommended surface speed (peripheral velocity) in the appropriate literature (Table 1):

$V \approx 250$ ft/min

Next, calculate the spindle speed from Equation 2:

N *[rpm]* $= 12 \times V / (\pi \times D)$
$= 12$ in/ft $\times$ 250 ft/min / ($\pi \times 1$ in/rev)
= **950 rpm (ans)**

Finally, calculate the feed rate using Equation 3, given that the appropriate initial feed per tooth (chipload) is $f_t \approx 0.008$ in/tooth (from Table 3):

f *[in/min]* $= N$ *[rpm]* $\times f_t$ *[in/tooth]* $\times m$ *[teeth/rev]*
= 950 rev/min $\times$ 0.008 in/tooth $\times$ 4 teeth/rev
= **30 in/min (ans)**

Note that these speed and feed values are guidelines assuming proper (flooded) lubrication, workpiece stiffness and depth of cut. When applying oil manually (as in the lab), scale the feed and speed back to 60%, so ***N*** **= 570 rpm and** ***f*** **≈ 18 in/min (final ans)**.