CNC LATHE GAGE SPEEDS AND FEEDS CALCS

Introduction

This document calculates the speeds and feeds for the cutting tools used in the Advanced Manufacturing CNC Lathe Gage Tutorial. The guiding calculations and underlying explanations come directly from the Speeds & Feeds resources link under the Advanced Manufacturing section of the DML website, as well as the comprehensive course notes on this topic, so refer to those or ask Mike if you have questions about how these calculations are performed.

Quick Links

Drilling and Milling Speeds and Feeds Equations

Tool 1: **0.166" HSS drill bit**
Tool 2: **0.089" HSS drill bit**
Tool 3: **4-40 UNC HSS tap**
Tool 4: **0.25" carbide endmill**
Tool 5: **0.5" carbide endmill**
Tool 6: **2" facemill with 6 carbide inserts**
Tool 7: **3/8" chamfer mill**

Surface Speeds Table
Drilling Feed Rates Table
Milling Feed Rates Table
Machinability of Plastics

Drilling and Milling Speeds and Feeds Equations [RETURN]

The governing equations are summarized below.

N [rpm] = 12 [in/ft] × V [sfm] / (π × D [in/rev]) ***Equation (1)***

where

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

f [in/min] = N [rpm] × f_r [in/rev] ***Equation (2) FOR DRILLING OPS***

f [in/min] = N [rpm] × f_t [in/tooth] × m [# of teeth] ***Equation (3) FOR MILLING OPS***

where

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

Recommended Surface Speeds for Common Materials [RETURN]

The table below contains a recommended surface speeds for common materials when using DML equipment. These values are conservative because our primary goal is fostering a safe learning environment (for our users and our tools!), not trying to squeeze every second out of each operation.

Material	Recommended HSS Speed, V [surface ft/min]
Acetal (Delrin)	250
Aluminum and its alloys	250
Brass (360 high machining)	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 work hardening	20
Titanium alloys	20
* multiply surface speeds in table by 2.5 for carbide cutting tools *	
* multiply surface speeds in table by 1.5 - 2.0 for HEM / HSM toolpaths *	

Recommended Feed Rates for 2 Flute HSS & Carbide Drills [RETURN]

Drill Diameter [in]	Recommended Feed per Tooth, 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 0.016
1” and over	0.016 and up
* multiply feed values in table by 0.5 for difficult to machine materials, flexible toolholding or workpieces, or lighter-duty machines*	
* consult manufacturer’s data for more accurate carbide drill feedrates *	

A general rule of thumb for materials which are strong enough to support the drilling process is that f_r is between 1 - 3% of the drill diameter, depending on the material strength.

Recommended Feed Rates for HSS and Carbide Endmills [RETURN]

Material	Recommended Feed per Tooth, f_t [in/tooth]

Acetal (Delrin)	0.008 to 0.030
Aluminum and its alloys	0.004 to 0.012
Brass (360 high machining)	0.004 to 0.012
Bronze (high tensile)	0.002 to 0.006
Cast Iron (soft)	0.002 to 0.006
Cast Iron (medium hard)	0.002 to 0.006
Cast Iron (hard chilled)	0.0015 to 0.004
Hastelloy	0.0015 to 0.004
Inconel	0.0015 to 0.003
Magnesium and its alloys	0.004 to 0.012
Monel	0.0015 to 0.003
High nickel steel	0.002 to 0.004
Mild steel (.2-.3 C)	0.002 to 0.008
Steel (.4-.5 C)	0.002 to 0.006
Tool steel	0.0015 to 0.004
Forgings	0.0015 to 0.004
Steel alloys (300-400 Brinell)	0.0015 to 0.004
Heat Treated Steels	
35-40 Rockwell C	0.0015 to 0.003
40-45 Rockwell C	0.0015 to 0.003
45-50 Rockwell C	0.0015 to 0.003
50-55 Rockwell C	0.001 to 0.002
stainless steel (free machining)	0.002 to 0.006
stainless steel (work hardening)	0.0015 to 0.004
Titanium alloys	0.0015 to 0.004
* table values are typical for cutters ranging from ½” to 1-1/2” in size *	

Speeds and Feeds Calcs for CNC Lathe Gage Demo Tools [RETURN]

Tool 1: Calculate the speed and feed for a 0.166" HSS drill bit in ½" thick aluminum blank on the TM-2 CNC milling machine in the lab. [RETURN]

First, lookup the recommended surface speed in Table 1 ($V \approx 250$ ft/min) and calculate the spindle speed from Equation 2:

$N\ [rpm] = 12 \times V / (\pi \times D)$

$= 12\ in/ft \times 250\ ft/min / (\pi \times 0.166\ in/rev)$

$\approx$ **5752 rpm**

Next lookup the recommended feed per revolution for the drill bit in Table 2 ($f_r \approx 0.0026$ in/rev) and calculate the feed rate using Equation 3:

$f\ [in/min] = N\ [rpm] \times f_r\ [in/rev]$

$= 5752\ rev/min \times 0.0026\ in/rev$

$\approx$ **15.0 in/min**

Scale Back for Non-ideal Conditions:

Note that these speed and feed values are guidelines assuming adequate (flooded) lubrication, machine and workpiece stiffness, and drill depth less than 3 drill diameters (0.498"). When drilling more than 3 drill diameters, scale the speed back 50%, so ***N = 2876 rpm and f = 7.5 in/min.*** When applying oil manually or via MQL (minimum quantity lubrication) as with the TM-2 in the lab, scale the feed and speed back to 60%, so ***N* = 1725 rpm and *f* = 4.5 in/min (final answer)**.

TIP: Recommended peck depth when drilling less than 3xD (e.g. 3 drill diameters) with flooded coolant is one drill diameter (0.166"). Scale it 50% for drilling > 3xD deep, and another 50% for applying oil manually, or using MQL, which in this case would result in a ***peck depth of 0.042".***

Tool 2: Calculate the speed and feed for a 0.089" HSS drill bit drilling 0.25" deep in an aluminum blank on the TM-2 CNC milling machine in the lab. [RETURN]

First, lookup the recommended surface speed in Table 1 ($V \approx 250$ ft/min) and calculate the spindle speed from Equation 2:

$N\ [rpm] = 12 \times V / (\pi \times D)$

$= 12\ in/ft \times 250\ ft/min / (\pi \times 0.089\ in/rev)$

$\approx$ **10730 rpm**

Next lookup the recommended feed per revolution for the drill bit in Table 2 ($f_r \approx 0.0014$ in/rev) and calculate the feed rate using Equation 3:

f [in/min] = N [rpm] × f_r [in/rev]
= 10,730 rev/min × 0.0014 in/rev
≈ **15.0 in/min**

Scale Back for Non-ideal Conditions and Spindle Speed Limitations:
Note that these speed and feed values are guidelines assuming adequate (flooded) lubrication, machine and workpiece stiffness, and drill depth less than 3 drill diameters (0.267”). When applying oil manually or via MQL (minimum quantity lubrication) as with the TM-2 in the lab, scale the feed and speed back to 60%, so ***N* = 6438 rpm and *f* = 9.0 in/min**. However, the maximum spindle speed on the TM-2 is only 4000 rpm, so **N = 4000 rpm and f = 5.6 in/min (final answer)**.

TIP1: Recommended peck depth when drilling less than 3xD (e.g. 3 drill diameters) with flooded coolant is one drill diameter, or when applying oil manually, or under low pressure, is 50% of drill diameter, or in this case, 0.083”.
~~*TIP2: When drilling deeper holes (> 3xD) without high pressure TSC (thru spindle coolant), reduce spindle speed an additional 50%.*~~ *This tip does not apply in this case.*

Tool 3: Calculate the speed and feed for a 4-40 UNC HSS tap in an aluminum blank on the TM-2 CNC milling machine in the lab. [RETURN]

First, lookup the major diameter of a 4-40 tap (D ≈ 0.112”), then lookup the recommended surface speed in Table 1 (V ≈ 250 ft/min) and calculate the spindle speed from Equation 2:
N [rpm] = 12 × V / (π × D)
= 12 in/ft × 250 ft/min / (π × 0.112 in/rev)
≈ **8525 rpm**

After observing the rigid tapping process a few times, it should make sense that at the bottom of each tap cycle (or peck, depending on how it’s programed), the spindle must decelerate, reverse direction, and reaccelerate as the tap is extracted from the threaded hole. These spindle speed accelerations must be **very closely** synchronized with corresponding Z-axis feedrate accelerations, or the result will be damaged threads or a broken tap. For this reason, only the best CNC machines with the highest-end servo motor amplifiers, position encoders, and machine controllers can successfully achieve these motions at these speeds. An entry-level machine like the Haas TM-2 likely ranks around 1 out of 10 when it comes to this synchronization capability, so the FASTEST it could successfully perform this operation would be around **850 rpm.**

Using a common pair of GO/NO GO thread gages, independent testing can be performed on any machine to determine the highest speed at which the synchronization is no longer able to achieve an allowable thread form. For reference, newer VF-series Haas VMCs may be capable of performing threading up to 1500 or 2000 rpm; the only way to know is to check.

To be safe when making prototypes parts in the lab (i.e. where tool survival is more important than cycle time, lol), select a few hundred rpm for the tapping speed. For our lathe gage, let's select **N = 400 rpm**, because I have confidence the machine can achieve the accelerations necessary to tap at this speed. Note we should have no problems with tool overheating since this selected spindle speed is so much lower than the computed maximum above (8525 rpm).

At this point it's important to realize a tap does not have a recommended chipload, but rather a synchronized feedrate at which the tap much be advanced in the Z-direction in relation to the programmed spindle speed. In the case of a 4-40 tap, **40 threads per inch** (tpi) is the thread pitch, which for a standard thread equates to [1 / 40] inch / thread, or **[1 / 40] inch / revolution.** Since we need to program the threading cycle with a parameter with units of inch / rev, we can use the previously selected spindle speed to compute the equivalent Z-axis feedrate:

$\mathbf{f_r}$ = [1/40] in / rev × 300 rev / min = **10 in / rev**

TIP: *For future reference, selecting the proper ratio of speeds and feeds for imperial taps can be simplified to just multiplying the tpi by a factor of 10 and using a f_r = 10 in/rev. For example, a 10-24 tap would result in N =240 rpm and f_r = 10 in/rev, and as long as both are scaled identically, you will have perfect commanded synchronization (i.e. you can program N = 360 rpm and fr = 15 in/rev with no issues).*

Tool 4: Calculate the speed and feed for a 0.25" diameter, 3 flute carbide endmill if peripheral milling in aluminum using the TM-2 CNC milling machine in lab. [RETURN]

First, lookup the recommended surface speed in Table 1 ($V \approx 250 \times 2.5 = 625$ ft/min) and calculate the spindle speed from Equation 2:

$N\ [rpm] = 12 \times V / (\pi \times D)$

$= 12\ \text{in/ft} \times 625\ \text{ft/min} / (\pi \times 0.25\ \text{in/rev})$

$\approx$ **9550 rpm**

*However, since the Haas TM-2 is limited to a top speed of 4000 rpm, we can see that the choice of carbide for this cutting tool was likely unwarranted. If we repeat the calculations for a HSS tool, the resulting spindle speed would be **N = 3820 rpm (final answer)**, which fits perfectly within our max speed limit of 4000 rpm for this machine.*

Next, lookup the recommended feed per tooth (chipload) in Table 3 ($f_t \approx 0.002$ in/tooth) and calculate the feed rate using Equation 3:

$f\ [in/min] = N\ [rpm] \times f_t\ [in/rev] \times m$

$= 3820\ \text{rev/min} \times 0.002\ \text{in/tooth} \times 3\ \text{teeth/rev}$

$\approx$ **23 in/min**

Note that these speed and feed values are guidelines assuming proper (flooded) lubrication, workpiece stiffness and depth of cut. When applying oil manually or using MQL (as in the lab), scale the feed and speed back to 60%, so ***N* = 2300 rpm and *f* ≈ 14 in/min (final answer)**. Note also this problem assumes we are peripheral milling versus plunge milling (if performing the latter, scale the previous spindle speed to 75% and feedback to 25%).

Tool 5: Calculate the speed and feed for a 1/2" diameter, 3 flute carbide endmill if peripheral and plunge cutting in aluminum using a CNC milling machine in lab. [RETURN]

First, lookup the recommended surface speed in Table 1 ($V \approx 625$ ft/min) and calculate the spindle speed from Equation 2:

$N_{peripheral}$ *[rpm] = 12 × V / (π × D)*

= 12 in/ft × 625 ft/min / (*π* × 0.5 in/rev*)*

≈ **4700 rpm**

We know from the previous examples, we must scale this back to **N = 4000 rpm** because of the max spindle speed of the TM-2.

Next, lookup the recommended feed per tooth (chipload) in Table 3 ($f_t \approx 0.004$ in/tooth) and calculate the feed rate using Equation 3:

$f_{peripheral}$ *[in/min] = N [rpm] × f_t [in/rev] × m*

= 4000 rev/min × 0.004 in/tooth × 3 teeth/rev

≈ **48 in/min**

TIP: Plunging should generally be performed at 75% of the speed and 25% of the feedrate of the calculated peripheral cutting parameters.

Based on the previous tip, calculate the same parameters for plunge milling using the noted scaling factors:

$N_{plunge} \approx 0.75 \times N_{peripheral}$

≈ 0.75 × 4700 rpm

≈ **3525 rpm**

$f_{plunge} \approx 0.25 \times f_{peripheral}$

≈ 0.25 × 56 in/min

≈ **14 in/min**

Note that these speed and feed values are guidelines assuming proper (flooded) lubrication, workpiece stiffness and depth of cut. When applying oil manually or using MQL (as in the lab), scale the feed and speed back to 60%, so **N = 2100 rpm and f = 8 in/min (final answer).**

Tool 6: Calculate the speed and feed for a 2" diameter, 6 flute facemill with smallish-sized, expensive carbide inserts if peripheral milling aluminum using the TM-2 CNC milling machine in lab. [RETURN]

First, lookup the recommended surface speed in Table 1 ($V \approx 625$ ft/min) and calculate the spindle speed from Equation 2:

$N_{peripheral}$ *[rpm] = 12 × V / (π × D)*

= 12 in/ft × 625 ft/min / (*π × 2* in/rev*)*

≈ **1200 rpm**

Next, lookup the recommended feed per tooth (chipload) in Table 3. Since the facemill uses fairly small and expensive inserts, select a conversative chipload from the table ($f_t \approx 0.006$ in/tooth) and calculate the feed rate using Equation 3:

$f_{peripheral}$ *[in/min] = N [rpm] × f_t [in/rev] × m*

= 1200 rev/min × 0.006 in/tooth × 6 teeth/rev

≈ **43 in/min**

Note that these speed and feed values are guidelines assuming proper (flooded) lubrication, workpiece stiffness and depth of cut. When applying oil manually or using MQL (as in the lab), scale the feed and speed back to 60%, so **N = 720 rpm and f = 25 in/min (final answer).**

Tool 7: Calculate the speed and feed for a 3/8" diameter 4 flute carbide chamfer mill used on an aluminum workpiece in the TM-2 CNC milling machine. [RETURN]

First, lookup the recommended surface speed in Table 1 ($V \approx 625$ ft/min) and calculate the spindle speed from Equation 2:

N [rpm] = 12 × V / (π × D)

= 12 in/ft × 625 ft/min / (*π* × 0.375 in/rev*)*

≈ **6400 rpm**

We know from the previous examples, we must scale this back to **N = 4000 rpm** because of the max spindle speed of the TM-2.

Next, lookup the recommended feed per tooth (chipload) in Table 3 ($f_t \approx 0.003$ in/tooth) and calculate the feed rate using Equation 3:

$f_{peripheral}$ *[in/min] = N [rpm] × f_t [in/rev] × m*

= 4000 rev/min × 0.003 in/tooth × 4 teeth/rev

≈ **48 in/min**

Note that these speed and feed values are guidelines assuming adequate (flooded) lubrication and workpiece stiffness. When applying oil manually (as in the lab), scale the feed and speed back to 60%, so ***N* = 2400 rpm and *f* = 29 in/min (final answer)**.