



**Industrial Technology  
Virtual Learning**

**General Metals  
Speed and Feed**

**April 10<sup>th</sup>, 2020**



# General Metals

## Speed and Feed

April 10<sup>th</sup>, 2020

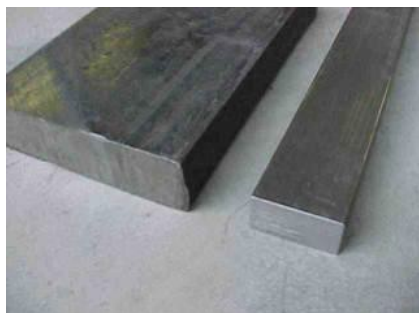
Objective: Students will be able to calculate proper speed and feed for drills and end mills used on the milling machines



# Calculating Speeds and Feeds

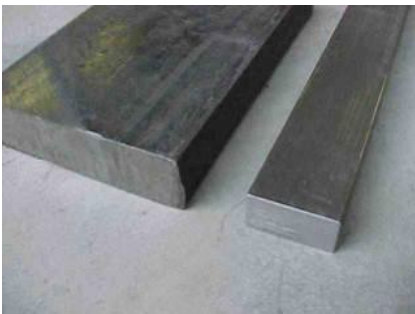
	Cutting Speed (Surface feet per minute)	Feed Rate (Chip load per tooth)
Aluminum	<b>300</b>	<b>.012</b>
Brass	<b>200</b>	<b>.010</b>
Low Carbon Steel	<b>100</b>	<b>.007</b>

*These are examples: Not to  
be used for actual machining*



# Calculating Speeds and Feeds

Can the same RPM for a drill bit be used for drilling wood and metal?



# Calculating RPMs

All RPM calculations are based on a value known as Constant Surface Speed (C.S.S.)



# Calculating RPMs

To calculate RPM's, the formula is:

$$\text{RPM} = 3.82 \times \text{CSS} / \text{Dia.}$$

CSS is the Constant Surface Speed in Surface Feet per Minute.

Dia. is the diameter of the cutting tool you are using (or the diameter of the workpiece you are cutting on a lathe).

# Constant Surface Speed

CSS: Rate at which material passes the cutting edge (or the rate at which the cutting edge passes the material).



# Constant Surface Speed

Also referred to as Surface Feet per Minute or SFM because it is expressed in surface feet per minute.

The following web site will be used for all information presented.

[https://www.usi.edu/science/engineering/machasst/cutinfo/spd\\_fed.htm](https://www.usi.edu/science/engineering/machasst/cutinfo/spd_fed.htm)



# Additional Resources

The following are additional resources that could also be used.

These resources were not used for this information.

Speed and Feed is not the same for all cutters. Advances in cutter technology has made speed and feed more specific to the manufacture of the cutter. For accurate information, always contact the manufacture of the cutting tool.

<https://littlemachineshop.com/reference/cuttingspeeds.php>

<https://www.the-carbide-end-mill-store.com/Feeds-and-Speeds.html>

<https://www.lakeshorecarbide.com/lakeshorecarbidecomspeedandfeedcharts.aspx>

[http://www.harveytool.com/cms/SpeedsFeeds\\_228.aspx](http://www.harveytool.com/cms/SpeedsFeeds_228.aspx)

# Where do you find CSS?

Use the recommended CSS found in the tooling manufacturers catalog or website. More information can be found in the Machinery Handbook.

Please use this resource for calculating speed and feed used in this lesson.

[https://www.usi.edu/science/engineering/machasst/cutinfo/spd\\_feed.htm](https://www.usi.edu/science/engineering/machasst/cutinfo/spd_feed.htm)

They have determined these values based on their own calculations as well as trial and error.

# Calculating RPMs



If you were drilling a  $5/8''$  (.625) dia. hole in ordinary standard bronze using a high speed steel drill, here's how you would calculate the RPMs.

1: Refer to the web site to determine the CSS.

(Refer to the ft/min column under the Drilling column from the Speed and Feed web site before).



# Calculating RPMs

2: Fill in the variables in the formula:

$$\text{RPM} = 3.82 \times 175 / .625 \quad (\text{175 is the midpoint between 100 and 250})$$

3: Perform the calculations:

$$\text{RPM} = 668.5 / .625 = 1069$$

Realize that a small diameter cutter or workpiece will have a higher RPM than that of a larger workpiece or cutting tool

# Calculating RPMs (practice)

Calculate the RPMs for the following processes  
(Note – Use midpoint of the suggested cutting speeds for your calculations):

- 1: Drilling a  $3/8$ " hole in aluminum using a high speed steel drill.
- 2: Drilling a  $1/8$ " hole in Magnesium using a high speed steel drill.
- 3: Milling steel, low C with a  $7/16$ " dia. high speed end mill.
- 4: Milling soft cast iron with a  $5/16$ " dia. High speed end mill.



## Calculating RPMs (practice)

Calculate the RPMs for the following processes:

$$1: (3.82 \times 425 \text{ sfm}) / .375" = 4329 \text{ RPM}$$

$$2: (3.82 \times 475 \text{ sfm}) / .125" = 14516 \text{ RPM}$$

$$3: (3.82 \times 82 \text{ sfm}) / .4375" = 715 \text{ RPM}$$

$$4: (3.82 \times 70 \text{ sfm}) / .3125" = 855 \text{ RPM}$$

# Calculating Feeds

If programming speeds and feeds on a CNC machine, feed is often expressed in inches per minute (ipm).

To calculate feed in ipm, the formula is:

Feed in ipm = RPM x ipr.

ipr = inches per revolution, and is also found in the manufacturers catalog.

# Calculating Feeds

To calculate feed in ipm for our original example (drilling a 5/8" hole in standard bronze using a high speed steel drill), we would multiply the RPM x ipr from the chart.

IPR for a .625 dia. drill would be .012 (proportional value of .007 to .017 found in the Feed inches/rev column at the bottom of the web site using the 1/2" to 1" drill size).

Feed in ipm would be calculated:  $1069 \text{ RPM} \times .012 \text{ ipr} = 12.83 \text{ ipm}$ .



# Calculating Feeds



Calculate the feed in ipm for the two drilling examples from the previous slide;

1: Drilling a  $3/8$ " hole in aluminum using a high speed steel drill.



2: Drilling a  $1/8$ " hole in Magnesium using a high speed steel drill.



# Calculating Feeds

1:  $4329 \text{ RPM} \times .0055 \text{ ipr} = 23.81 \text{ ipm}$

2:  $14516 \text{ RPM} \times .00 \text{ ipr} = 14.516 \text{ ipm}$

Note: .001 ipr was chosen in problem #2 due to the frailness of the small drill diameter, better to use a smaller feed amount than to break the drill bit.

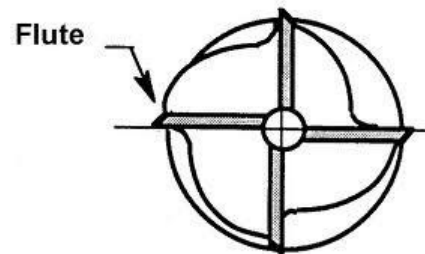


# Calculating Feeds

You'll notice that feeds given on your drilling chart are already in ipr but the feeds on your milling chart are given in inches per tooth.

When feed is given in inches per tooth (ipt), you must calculate ipr by multiplying the number of teeth on the tool you are using by the ipt.

$\text{Ipr} = \text{ipt} \times \text{number of teeth}$



# Calculating Feeds

Calculate the feed in ipm for the two milling examples from the previous slide using the midpoint ipt;

3: Milling steel, low C with a  $7/16''$  dia. carbide end mill (3 cutting flutes).

4: Milling soft cast iron with a  $5/16''$  dia. carbide end mill (4 cutting flutes).



## Calculating Feeds

$$3: 715 \text{ RPM} \times (.003 \text{ ipt} \times 3 \text{ teeth}) = 6.44 \text{ ipm.}$$

$$4: 855 \text{ RPM} \times (.0035 \text{ ipt} \times 4 \text{ teeth}) = 11.97 \text{ ipm.}$$

You are now ready to complete the  
Calculating Speeds and Feeds  
Written Assignment.

Determining Approximate Milling Cutter Speed & Feed  
For the Following Examples

RPM Formula:  $RPM =$

Feed Formula:  $Feed (F) =$

1. 1" dia. Drill Bit  
HSS  
Copper  
Speed (RPM'S) \_\_\_\_\_  
Feed (F) \_\_\_\_\_
2. 1/4" dia. Drill Bit  
HSS  
Aluminum  
Speed (RPM'S) \_\_\_\_\_  
Feed (F) \_\_\_\_\_
3. 1.125" Drill Bit  
HSS  
Stainless Steel, Free Machining  
Speed (RPM'S) \_\_\_\_\_  
Feed (F) \_\_\_\_\_
4. 3/4" Drill Bit  
HSS  
Cast Iron, Hard  
Speed (RPM'S) \_\_\_\_\_  
Feed (F) \_\_\_\_\_
5. 3/4" End mill  
HSS  
4 Flutes  
Cast Iron, Soft  
Speed (RPM'S) \_\_\_\_\_  
Feed (F) \_\_\_\_\_
6. 1/4" End mill  
HSS  
2 Flutes  
Stainless Steel 304  
Speed (RPM'S) \_\_\_\_\_  
Feed (F) \_\_\_\_\_
7. 1/2" End mill  
HSS  
3 Flutes  
Aluminum  
Speed (RPM'S) \_\_\_\_\_  
Feed (F) \_\_\_\_\_
8. 1" End mill  
HSS  
6 Flutes  
Plastic  
Speed (RPM'S) \_\_\_\_\_  
Feed (F) \_\_\_\_\_