Speeds, Feeds, & DOC
Turning - Presentation Overview

• Turning
  – RPM vs. Surface footage
    • Definition
    • How to calculate
    • When and Why to use
    • Material machinability ratings
  – Feed rate
    • Under or Over feeding and its effect
    • Feed rate on materials
    • Effect on tools
    • Calculating material removal
Definitions

• Revolutions per minute (RPM)
  – It is a measure of the frequency of a rotation.
  – It represents the number of turns completed in one minute around a fixed axis. It is used as a measure of rotational speed of a mechanical component.

• Surface Speed (sfm or m/min)
  – It is the speed difference (relative velocity) between the cutting tool and the surface of the workpiece it is operating on. It is expressed in units of distance along the workpiece surface per unit of time, typically surface feet per minute (sfm) or meters per minute (m/min).
MPH vs. SFM

SFM is similar to MPH. Materials have speed limits!

"How fast are you going?" is about your SFM not RPM
Questions

• How many of you have specific RPM’s you always use to perform certain tasks?

  – I always run my part-off tool at 1000RPM
  – We run this spot drill at 1500RPM.
  – ½” end mills should be run at 750RPM

• WHY?
Questions

• Examples
  – I always run my part-off tool at 1000RPM
    • On ½” Ø material this is 130 SFM
    • ¾” Ø it’s 200 SFM
    • 1” Ø it’s 260 SFM
  – We run this 90° spot drill at 1500RPM
    • Depth controls SFM
    • 1/4” deep for 1/2” Ø chamfer is 200 SFM
    • 3/8” deep for 3/4” Ø chamfer is 300 SFM
  – ½” end mills should be run at 2000RPM
    • This is 260 SFM – material controls if this proper speed

• Where did these numbers come from?
Formulas

Where do they come from?
What do they mean?

\[ RPM = \frac{Speed}{Circumference} = \frac{Speed}{\pi \times Diameter} \]

and using the same example as above

\[ RPM = \frac{100 \text{ ft/min}}{\pi \times 10 \text{ inches} \left( \frac{1 \text{ ft}}{12 \text{ inches}} \right)} = \frac{100}{2.62} = 38.2 \text{ revs/min} \]

and using the same example as above

\[ RPM = \frac{30 \text{ m/min}}{\pi \times 10 \text{ mm} \left( \frac{1 \text{ m}}{1000 \text{ mm}} \right)} = \frac{1000 \times 30}{\pi \times 10} = 955 \text{ revs/min} \]
Math explained

\[ \text{RPM} = \frac{\text{Speed}}{\text{Circumference}} \quad \rightarrow \quad \text{Circumference} = \pi \times \text{diameter} \quad \rightarrow \quad \frac{\text{Speed}}{\pi \times \text{Diameter}} \]

\[ \frac{\text{Speed}}{\pi \times \text{Diameter}} \quad \rightarrow \quad \text{Speed is given in feet/min and diameter is given in inches so we need to convert to like values} \quad \rightarrow \quad \frac{12 \times \text{Speed}}{\pi \times \text{Diameter}} \]

\[ \frac{12 \times \text{Speed}}{\pi \times \text{Diameter}} \quad \rightarrow \quad \text{You cannot divide by two numbers so this needs to be simplified} \quad \rightarrow \quad \frac{12 \times \text{Speed}}{\pi \times \text{Diameter}} \times \frac{\pi}{\pi} \]

\[ \frac{\frac{12}{\pi} \times \text{Speed}}{\text{Diameter}} \quad \rightarrow \quad 12 / \pi = 3.82 \quad \rightarrow \quad \frac{3.82 \times \text{Speed}}{\text{Diameter}} \]
Formulas simplified

To find RPM when SFM is known:

\[ RPM = \frac{3.82 \times Speed}{Diameter} \]

INVERSE

To find SFM when RPM is known:

\[ Speed = \frac{RPM \times Diameter}{3.82} \]
Calculation example - together

- Rough turn OD
  - Diameter being created = 1.750”
  - RPM currently being run = 2000
  - What SFM is being run?

\[
Speed = \frac{RPM \times Diameter}{3.82}
\]

1.750 \times 2000 = 3500
3500 / 3.82 = 916 SFM
Calculation example - together

• ID Groove
  – Bore diameter = 1.25”
  – Groove diameter = 1.900”
    • Depth of groove/side = .325”
  – RPM currently being run = 1250
  – What SFM is being run?

1.25 x 1250 = 1562.5
1562.5 / 3.82 = 409 SFM

1.900 x 1250 = 2375
2375 / 3.82 = 622 SFM
When & Where to use

- **When & Where**
  - Anytime there will be diameter changes
    - Even the smallest variation can affect tool performance
      - Turning multiple diameters
      - ID & OD Grooving
      - Facing off or Parting-off (even to center)
        » Word of warning your spindle will max out
        » Work piece could come lose
        » Be sure to use spindle limiter
When & Where to NOT use

• No advantage
  – Anytime tool is at center
    • Drilling
    • Tapping
    • Reaming
  – Threading
  – Turning of a single diameter
    • You can use here, and it is not a bad idea
Material machinability

• What is Machinability?
  – The term **machinability** refers to the ease with which a metal can be cut permitting the removal of the material.
  – Of course factors that typically improve a material's performance often degrade its machinability
  – Materials are considered to be **free machining** when they require little power to cut, can be cut quickly, easily obtain a good finish, and do not wear the tooling much
Material machinability – base line

• The “machinability chart” is created around an industry standard
• All values shown are directly related as a percentage to this standard material
  – B1112 @ 160 HB is 1.00 or 100% (180SFM)
    • Materials with lower values will be more difficult to machine, and high values will be easier
Material machinability – sample chart

<table>
<thead>
<tr>
<th>Material</th>
<th>Rating</th>
<th>100% Rating</th>
</tr>
</thead>
<tbody>
<tr>
<td>3340</td>
<td>220</td>
<td>0.45</td>
</tr>
<tr>
<td>3450</td>
<td>197</td>
<td>0.45</td>
</tr>
<tr>
<td>4130 (Castings)</td>
<td>175</td>
<td>0.35</td>
</tr>
<tr>
<td>4130</td>
<td>183</td>
<td>0.65</td>
</tr>
<tr>
<td>4140</td>
<td>190</td>
<td>0.55</td>
</tr>
<tr>
<td>4140 (Leaded)</td>
<td>187</td>
<td>0.70</td>
</tr>
<tr>
<td>4145</td>
<td>200</td>
<td>0.55</td>
</tr>
<tr>
<td>4340 (100% pearlitic)</td>
<td>221</td>
<td>0.45</td>
</tr>
<tr>
<td>4340 (Spheroidized)</td>
<td>206</td>
<td>0.65</td>
</tr>
<tr>
<td>4340 (Castings)</td>
<td>300</td>
<td>0.25</td>
</tr>
<tr>
<td>4620</td>
<td>170</td>
<td>0.65</td>
</tr>
<tr>
<td>4640</td>
<td>187</td>
<td>0.55</td>
</tr>
<tr>
<td>4815</td>
<td>183</td>
<td>0.55</td>
</tr>
<tr>
<td>5120</td>
<td>191</td>
<td>0.65</td>
</tr>
<tr>
<td>6130</td>
<td>183</td>
<td>0.55</td>
</tr>
<tr>
<td>6135</td>
<td>190</td>
<td>0.55</td>
</tr>
<tr>
<td>6180</td>
<td>207</td>
<td>0.40</td>
</tr>
<tr>
<td>8030 (Castings)</td>
<td>175</td>
<td>0.45</td>
</tr>
<tr>
<td>8430 (Castings)</td>
<td>180</td>
<td>0.40</td>
</tr>
<tr>
<td>8620</td>
<td>194</td>
<td>0.60</td>
</tr>
</tbody>
</table>

B-1112 | 160 | 1.00 |

- 100% rating
Feed rate

• Running tools within the proper feed range is critical

• Under or over feeding can have catastrophic results

• All tools are designed to work within a known range, as that was initial design intent

© Horn USA, Inc.
Feed rate - video
Feed rate – under feeding

- Tools will typically have a honed edge
  - This hone helps protect the cutting edge
- Always feed faster than size of hone
- Under feeding a tool is the same as rubbing
  - You’re not cutting, you’re displacing
  - Need to feed fast enough to engage chip former
- Same can hold true for corner radii
  - Engage at least 75%, prefer to bury
- DOC can affect tool life as well
  - Must be greater than hone size
Feed rate – over feeding

• Over feeding a tool can cause chip issues
  – Chip formers are designed to accept a specific chip flow
• Tools can only handle so much pressure
  – If feed rate is too excessive tools can have mechanical failure
• DOC can play into the ability to form the chip
Feed rate – video (positive geometry)
Feed rate - video (aggressive geometry)
Feed rate – Sweet spot

Sweet Spots for Feeds and Speeds

- Feeding too much chipload: Tool Breakage
- MRR
- Best tool life
- Moderate MRR
- Moderate surface finish
- Surface finish
- Older machines may prefer HSS tooling!
- Feeding too slow: Rubbing / Poor tool life

© Horn USA, Inc.
Feed rate – other wear images

**Flank wear**
Preferred wear as it is the most predictable

**Crater wear**
Localized to rake side
Chemical reaction

**Built up edge (BUE)**
Pressure welding of chips
Usually caused by too low cutting speeds

**Thermal cracks**
Rapid temperature changes
Interrupted cut, or coolant issues

**Notch wear**
Localized to rake side
DOC notch / pressure welding

**Plastic Deformation**
Tool material is softened
Cutting temps to high
Chip thickness with lead angles

• Lead angle tools
  – Variations can affect chip thickness
    • May allow for increase in feed rate, or allow better chip control
  – Does create longer cross-section chip length
Cubic volume material removal - calc

- Calculating in³/min material removal rate
  - Can be helpful when comparing different tool/processes
  - Ultimate affect to speed, feed, and DOC changes
  - Assist in tool life comparisons

CUBIC INCHES PER MINUTE METAL REMOVAL RATE (in³/min)

MMR = SFM x 12 x IPR x DOC

© Horn USA, Inc.
Sample

- **Examples**
  - **OD turning**
    - Speed = 550 SFM
    - Feed = .012 IPR
    - DOC = .075”/side
  - $550 \times 12 \times .012 \times .075 = 5.94\text{in}^3/\text{min}
  - Same as above w/10% drop in speed and 10% increase in feed
    - $495 \times 12 \times .0132 \times .075 = 5.88\text{in}^3/\text{min}$

**CUBIC INCHES PER MINUTE METAL REMOVAL RATE (in}^3/\text{min})**

$$\text{MMR} = \text{SFM} \times 12 \times \text{IPR} \times \text{DOC}$$
Calculate together

• **Situation**
  – You are testing a new insert against the one you are currently running. You are told that your current speed is way to fast, but this new tool can handle a greater DOC and heavier feed so it will be a wash. Lets compare the MMR values.
  – **Current**
    • Speed = 850 SFM
    • Feed = .008 IPR
    • DOC = .05”/side
  – **Suggested**
    • Speed = 510 SFM (40% decrease)
    • Feed = .012 IPR (50% increase)
    • DOC = .055”/side (10% increase)

• **Which will be faster?**
  – $850 \times 12 \times .008 \times .05 = 4.08\text{in}^3/\text{min}$
  – $510 \times 12 \times .012 \times .055 = 4.04\text{in}^3/\text{min}$
Milling - Presentation Overview

• Milling
  – Surface footage
    • How to calculate
    • Partially engaged tools and calculation
  – Feed rate
    • Chip thinning
    • Engagement
    • Approach option
    • Material removal rate / calculation
Formulas - revisit

Where do they come from?  
What do they mean?

\[ RPM = \frac{Speed}{Circumference} = \frac{Speed}{\pi \times Diameter} \]

and using the same example as above

\[ RPM = \frac{100\text{ ft/min}}{\pi \times 10\text{ inches} \left(\frac{1\text{ ft}}{12\text{ inches}}\right)} = \frac{100}{2.62} = 38.2\text{ revs/min} \]

and using the same example as above

\[ RPM = \frac{30\text{ m/min}}{\pi \times 10\text{ mm} \left(\frac{1\text{ m}}{1000\text{ mm}}\right)} = \frac{1000 \times 30}{\pi \times 10} = 955\text{ revs/min} \]
Formulas simplified

To find RPM when SFM is known:

\[
\text{rpm} = \frac{12 \times \text{ft/min}}{\pi \times D}
\]

\[
\text{rpm} = \frac{3.82 \times \text{ft/min}}{D}
\]

INVERSE

To find SFM when RPM is known:

\[
\text{ft/min} = \frac{D \times \text{rpm}}{3.82}
\]
Calculation example - together

• ½” diameter HSS drill
  – Book suggested speed = 250 SFM
  – What RPM should be run?

\[
r/\text{min} = \frac{3.82 \times \text{ft/min}}{D}
\]

\[
3.82 \times 250 = 955
\]
\[
955 / .500 = 1910 \text{ RPM}
\]
Calculation example - together

- 4” diameter 90° insert face mill
  - Book suggested speed = 600
  - What RPM should be run?

\[
\text{r/min} = \frac{3.82 \times \text{ft/min}}{D}
\]

\[
3.82 \times 600 = 2292 \\
2292 / 4 = 573 \text{ RPM}
\]
Calculation for partially engaged tool

Effective cutting diameter $d_{\text{eff}}$ of $\beta = 0^\circ$

$$d_{\text{eff}} = 2 \sqrt{d \cdot a_p - a_p^2} \quad (\text{mm})$$

Example:

$d = \varnothing 12 \text{ mm}; a_p = 1.2 \text{ mm}; \beta = 0^\circ$  

$\quad \rightarrow \quad d_{\text{eff}} = 7.2 \text{ mm}$
Calculation for partially engaged tool

Effective cutting diameter $d_{\text{eff}}$ of $\beta = +20^\circ$

$$d_{\text{eff}} = d \cdot \sin [\beta + \arccos \left(\frac{d - 2a_p}{d}\right)]$$

Example:

$d = \varnothing 12$ mm; $a_p = 1.2$ mm; $\beta = +20^\circ$

$$d_{\text{eff}} = 12 \cdot \sin [20^\circ + \arccos \left(\frac{12 - 2 \cdot 1.2}{12}\right)]$$

$d_{\text{eff}} = 10.05$ mm
Calculation for partially engaged tool

Effective cutting diameter $d_{\text{eff}}$ of $\beta = -20^\circ$

$$d_{\text{eff}} = d \cdot \sin \left[ \arccos \left( \frac{d - 2a_p}{d} \right) - \beta \right]$$

Example:

$d = \varnothing 12 \text{ mm}; a_p = 1.2 \text{ mm}; \beta = -20^\circ$

$$d_{\text{eff}} = 12 \cdot \sin \left[ \arccos \left( \frac{12 - 2 \cdot 1.2}{12} \right) -20^\circ \right]$$

$d_{\text{eff}} = 3.48 \text{ mm}$
Calculation for partially engaged tool
Chip thinning - calculation

Radial chip thinning has been around for decades. It is the observation that when the stepover is equal to or greater than 50% of the tool's width the thickness of the chip at a given feedrate remains constant.

However, when the step-over is less than 50%, the chip width becomes progressively smaller as the stepover decreases.

FIG 1

© Horn USA, Inc.
Calculation – linear milling

- Words of caution
  - Be sure you understand what published feed values are offered from the manufacturer
    - $f_z$ is desired chip thickness at entry
    - $h_m$ is desired chip thickness at center of cut
    - Some manufacturers publish multiplier factors

This is chip thinning formula

$$\text{IPT} \left( \frac{\sqrt{(D - a_e) \times a_e}}{r} \right)$$
Engagement calculations

- Examples
  - 4” Ø cutter @ 600 RPM
  - 8 effective w/ 25% stepover
  - suggested feed/tooth .004”
    - OLD method
      - \( \text{RPM} \times f_z \times Z = \text{feed/min} \)
      - \( 600 \times .004 \text{ IPT} \times 8 \text{ eff.} = 19.2 \text{ IPM} \)
    - Corrected method
      - \( \sqrt{(4-1) \times 1} = 1.732 \)
      - \( 1.732 \div 2 = .866 \)
      - \( .004 \div .866 = .0046 \)
      - \( 600 \times .0046 \times 8 = 22.1 \text{ IPM} \)

OVER A 15% INCREASE!
Engagement – circular interpolation

• During **internal** circle interpolations, the outside diameter of a milling cutter advances **faster** than the centerline of the tool.
  – This will produce a thicker chip than the calculated feed rate. (IPT)

• During **external** circle interpolations, the outside diameter of a milling cutter will advance **slower** than the centerline of the tool.
  – This will produce a thinner chip than the calculated feed rate. (IPT)

• When less than 1/2 of the diameter of the tool is engaged (radial) with the workpiece, there will also be **radial chip thinning effects**.
Approach engagement linear milling

- Angle of inclination can play a strong roll in tool performance
- This is only a sample from one tooling manufacturer

<table>
<thead>
<tr>
<th>Work Material</th>
<th>Appropriate E</th>
<th>Cutter dia. and ( a_e )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>(-42^\circ)</td>
<td>(a_e \approx \frac{2}{3} \delta D_c)</td>
</tr>
<tr>
<td>Cast iron</td>
<td>(-53^\circ)</td>
<td>(a_e \approx \frac{4}{5} \delta D_c)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Work Material</th>
<th>Appropriate E</th>
<th>Cutter dia. and ( a_e )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>(-30^\circ)</td>
<td>(a_e \approx \frac{3}{5} \delta D_c)</td>
</tr>
<tr>
<td>Cast iron</td>
<td>(-40^\circ)</td>
<td>(a_e \approx \frac{3}{4} \delta D_c)</td>
</tr>
</tbody>
</table>
Approach engagement circular

- Entry angle for internal and external grooving can has similar results to the inclination angle of milling cutters
- Smooth entry into the cut, or in thread milling proper matching to thread pitch is important
- 45 degrees is common entry angle

Ramp angle > 45°
Calculation for partially engaged tool
Cubic volume material removal - calc

- The **material removal rate**, typically expressed as Q in formulas, refers to the volume removed in the form of chips over a certain time during the material-removal machining process.

- In production technology, the material removal rate is a **significant factor** as it is used as the basis for measuring and rating the **productivity of tools and machines**.

- The aim of many developments in the field of tools and machine tools is to improve the material removal rate by **increasing** the **cutting speed** or **feed rate**.
Cubic volume material removal - calc

\[ Q = a_e \text{ [mm]} \times a_p \text{ [mm]} \times V_f \text{ [mm/min]} = \text{cm}^3/\text{min} \]

\[ Q = a_e \text{ [in]} \times a_p \text{ [in]} \times V_f \text{ [in/min]} = \text{in}^3/\text{min} \]
Sample

• Examples
  ➢ 4” Ø cutter @ 600 RPM
  ➢ 8 effective w/ 25% stepover (1” $a_e$)
  ➢ .250” DOC ($a_p$)
  ➢ 5” LOC
  ➢ Suggested feed/tooth .004”
  ➢ Programmed feed rate = 22.1 IPM (chip thinning)

$$(a_e) \ 1" \ x \ (a_p) \ .250" \ x \ (V_f) \ 22.1 \ \text{IPM} = 5.525 \text{in}^3/\text{min}$$
Calculate together

- Let’s incorporate a few things we’ve learned today
  - Calculation of proper RPM based on known SFM
  - Calculation of programmed feed using chip thinning
  - Calculation of MMR (material removal rate)
Calculate together

• Known values
  – Calculation of proper RPM based on known SFM
  – Calculation of programmed feed using chip thinning
  – Calculation of MMR (material removal rate)
Calculate together

• Known values
  – 2” diameter face mill cutter
  – 7 effective
  – \( a_e = 0.5” \)
  – \( a_p = .250” \) DOC
  – Recommend speed = 800 SFM
  – Recommend \( f_z \) at entry of cut = .008”
  – LOC = 5”
  – Material is 1018 steel
Calculate together

- **SFM** = \[ \frac{3.82 \times \text{ft/min}}{D} \]
  
  \[ 3.82 \times 800 / 2 = 1,528 \text{ RPM} \]

- **Programmed IPM** = \( \frac{\text{IPT}}{\sqrt{(D - a_e) \times a_e}} \)
  
  \[ \sqrt{(2 - .5) \times .5} = .866 \]
  
  \[ .866 \div 1 = .866 \]
  
  \[ .008 \div .866 = .0092 \]
  
  \[ 1528 \times .0092 \times 7 = 98.4 \text{ IPM} \]

- **MMR (material removal rate)** = \( a_e \times a_p \times V_f = \text{in}^3/\text{min} \)
  
  \[ .5 \times .25 \times 98.4 = 12.3 \text{ in}^3/\text{min} \]
Questions !??!