TROUBLESHOOTING AND MILLING STRATEGIES

Surface finish quality

If surface finish quality is sub-par or does not meet specifications

- Review/improve workpiece holding and tool holding
- Reduce feed rate (IPM)
- Increase cutting speed (RPM)
- Review tool selection / switch to tool better suited for finish requirements

Vibration

Excessive tool wear, poor surface finish, loud audible tool noise, and poor dimensional accuracy

- Review/improve workpiece holding and tool holding
- Increase feed rate (IPM)
- Decrease cutting speed (RPM)
- Review milling strategy.
  ie: Climb vs conventional milling, or WOC / DOC
- Review tool selection.
  ie: Number of teeth, helix angle, etc.

Chip congestion / Tool breakage

Insufficient chip evacuation, chipping on cutting edges, or edge build-up resulting in poor tool life or breakage

- Review delivery method of coolant / air
- Increase / adjust coolant flow
- Reduce feed rate (IPM)
- Review tool selection.
  ie: Number of teeth, helix angle, etc.

Feed rate adjustments for extended DOC

When extended DOC is used, adjustment of the feed rate may be necessary to avoid deflection, chatter, and poor surface finish

- if DOC = 1xD, use feed rate per recommended values
- if DOC = 2xD, reduce feed rate 50%
- if DOC = 3xD, reduce feed rate 75%
- Cutting speed (SFM) can remain unchanged up to DOC of 3xD

Feed rate adjustments for Ramping & Plunging

Feed rates should be adjusted for ramping and plunging applications

- On a 90 degree vertical plunge with center cutting endmill reduce feed rate 70%
- Ramping with ramp angles 1-15 degrees , use normal recommended feed rates
- Ramping with angles from 15-30 degrees, reduce feed rate by 30%
- Helical plunging reduce feed rate by 15% with .100-.200 DOC per revolution

Entry in Hard Material

Feed rates should be adjusted into material over 44 RC

- Arcing into the cut is always recommended if possible
- When arc entry is not possible, see recommendation for feed reduction below:
  - When WOC is 1xD, reduce feed 50% until tool is engaged
  - When WOC is .5xD, reduce feed 30% until tool is engaged
  - When WOC is .25xD, reduce feed 20% until tool is engaged

CONVERSIONS

Inch

SFM	Surface Feet per Minute	**SFM = RPM x D / 3.82**
RPM	Revolutions Per Minute	**RPM = 3.82 x SFM / DIA**
IPT	Inches Per Tooth	**IPT = IPM / (Z x RPM)**
IPM	Inches Per Minute	**IPM = IPT x Z x RPM**
CIM	Cubic Inches per Minute	**CIM = Feed x WOC x DOC**

D = Tool Diameter Z = Number of flutes
WOC = Width Of Cut DOC = Depth Of Cut

Metric

Vc	Surface Meters per minute	**Vc = 3.1416 x D x n / 1000**
n	Revolutions per minute	**n = Vc x 1000 / 3.1416 x D**
fz	Feed per tooth	**fz = vf / n x z**
Vf	Millimeters per minute	**vf = (n) x (z) x (fz)**

D = Tool Diameter (mm) Z = Number of flutes

Conversion Formulas

Inch to mm	**Inch number x 25.4**
mm to Inch	**mm number x .03937**

PROFILE THUMBNAILS:

SIDEWALL RADIUS | CIRCULAR INTERPOLATION | SLOTTING | PROFILE MILLING | SIDEWALL SHARP