

# Thinning Your Chips Can Fatten Your Profits!

## [What is Chip Thinning?](#)

## [How can we use chip thinning to make us more profitable?](#)

- **Chip Thinning - What is it?**

Lowering the LEAD ANGLE of a face mill, causes the chip being formed to be thinner than the IPT feedrate programmed into the machine. Therefore, when we run a 90° face mill versus a 45° face mill at the same Chip Load, the average chip thickness will be much less on the 45° face mill. In order to maintain the average chip thickness of .006" when using a 45° face mill, we would have to increase the feedrate of the 45° face mill. Another Advantage is that the horsepower draw is also less with the 45° face mill versus the 90° face mill running at the same speeds & feeds. As a direct result of these two anomalies we can run our 45° lead face mills faster than 90° face mills. This equates to faster metal removal rates & higher profits.

- **Feed Mills**

Feed Mills are designed to take the fullest advantage of the above theory. They do this by giving us the least amount of lead angle found in a milling cutter today, usually somewhere around 12°. This allows us to run very aggressive feedrates in order to maintain our average chip thickness with these cutters.

Feed mills were first introduced to the Mold & Die industry. With the help of these tools, roughing cycle times were cut by 50% or more. I have personally introduced these cutters in several shops, and have never left without the sale. I say this not because I am the world's greatest salesman, but because these cutters equate to unbelievable metal removal rates, which equates to higher profitability.

Here are some real life examples of what a Feed Mill can do!

### Example 1:

Customer was using a button cutter. **Operation** = Rough a die cavity **Material** was FX steel.

**Limiting factors** – **Max Feed rate** - 200 IPM

The customer was given a 2" feed mill. Because FX steel is somewhere around 32 Rc, the SFM was reduced in order to increase insert life.

Cutter Description	Cutter Diameter	No. of Inserts	Radial DOC	Axial DOC	SFM	RPM	CPT	IPM	In <sup>3</sup> /MIN
Button Cutter	3.00	6	2.100	0.060	800	1019	0.012	73.3	9.241
Feed Mill	2.00	4	1.200	0.080	600	1146	0.043	197.1	18.923

\* Even using a smaller cutter and reducing the SFM, we were able to cut the roughing cycle time in Half.

**Example 2:**

Customer was using a button cutter. **Operation** = Rough a die cavity **Material** was 4140 steel.

**Limiting factors** - The customer had an older machine & didn't want to exceed 50% of spindle load.

Cutter Description	Cutter Diameter	No. of Inserts	Radial DOC	Axial DOC	SFM	RPM	CPT	IPM	In <sup>3</sup> /MIN
Button Cutter	1.25	3	0.750	0.060	600	1834	0.012	66.0	2.970
Feed Mill	1.25	3	0.700	0.040	600	1834	0.043	236.5	6.623

\* We reduced the HP drain by taking a lighter depth of cut, & we were still able to easily out perform the button cutter. Cycle time was cut from 40 min to 18 min.

**Example 3:**

Customer was using a Solid Carbide End Mill. **Operation** = Rough a slot in a block, 1.67" wide x 3" deep x 3" long. **Material** was 1018 steel.

**Limiting factors** - **RPM** - none, **Feed rate** - none

Cutter Description	Cutter Diameter	No. of Flutes	Radial DOC	Axial DOC	SFM	RPM	CPT	IPM	In <sup>3</sup> /MIN
Solid Carbide End Mill	0.50	4	0.418	0.500	300	2292	0.0015	13.8	2.874
Feed Mill	1.00	2	0.835	0.060	900	3438	0.0420	288.8	14.468

\* The customer was initially losing money making these parts. His cycle time was cut from 5 min to less than 1 min by switching to a Feed Mill. With this kind of metal removal rate, the customer was finally able to make a profit on his parts.

- [Example: MRR of a 3" Feed Mill versus a 10" 45° Face Mill](#)

Look at the difference in size of the Feed Mill, yet it can easily attain the same MRR as the 10" Face Mill cutting Alloyed Steel.

Cutter Description	Cutter Diameter	No. of Inserts	Radial DOC	Axial DOC	SFM	RPM	CPT	IPM	In <sup>3</sup> /MIN
Feed Mill	3.00	5	2.100	0.098	600	764	0.079	301.8	62.106
45 Deg Face Mill	10.00	16	7.000	0.300	600	229	0.008	29.3	61.609

\* Most shops don't have the equipment to run a 10" Face Mill. The chart shows how you can achieve the same MRR as a 10" face mill, using a 3" Feed Mill.

- **High Feed Milling - Benefits**

As you can see by the examples above, Feed Mills work well no matter what industry you're in. Whether you are cutting & sinking Dies & Molds, or slotting a block, or just rough facing a part. If you are removing a lot of metal & you want to cut cycle time, think about Feed Mills.

Feed Mills are very versatile. They can remove more metal with less HP than conventional mills. Simply reduce the DOC & increase the feedrate to improve your MRR.

Feed Mills are designed to send the cutting forces into the bed of the machine, which helps with vibration dampening. This is an obvious advantage in long reach & unstable conditions.

Please don't hesitate to call your Sanders rep for a demo at your shop!