

Cutting Corners?

One look at ELMA's latest ad in ECN Magazine's January 2004 Issue and you'll realize that when they speak of cutting corners, it has nothing to do with finances or budget. Instead, the ELMA ad features a couple of panels being magnified to show the tightest corners in the industry. We know what you're thinking, "it's just an ad, how do I know these braggarts aren't just stretching the truth?" Well, we know because we sold them the machine to do it — a Datron router running a .3mm radius bit for the squarest corners and most precise edges. Now, that's something to brag about.

About ELMA: Elma is an innovator in the manufacture of electronic enclosures and passive electronic components. They have achieved a leading position in the CPCI, VME/VME64x, VXI, and Rugged COTS packaging markets.

It's Showtime!

In the last two issues of RPM, we covered some unique Datron features like the Z-Correction Probe™ and the Tool Management System™ — things that inspired us to stake a claim on the tagline "Smart CNC Machines". Now, at the EASTEC 2004 APEX event, May 25-27, you can stop by Booth #3347 in the Young Building and see for yourself just what makes us so smart — fact is, even Bobby Fisher has nothing over Datron! (See if you can figure out THAT teaser!)

DATRON Dynamics is a distributor for DATRON Electronics Makers of the awe-inspiring 60,000 RPM CNC Machine Centers
454 Route 13 • Milford, New Hampshire 03055 • USA
t 888.262.2833 • f 603.672.8067 • www.datrondynamics.com
© 2003. All rights reserved.

Tips: Feeds & Speeds

By Bill King, Vice President, Datron Dynamics

It has always been difficult for us to give exact formulas or precise data on machining parameters to our customers. There are so many variables involved when machining with high speed. However, here are some general guidelines for high speed machining with Datron tooling.

Generally you will receive best results regarding quality and cycle time by taking multiple shallow cuts quickly, compared to trying to machine in one pass slowly. This is all because of chip evacuation. As a general rule with high speed, the cutting depth in non-ferrous metal should be about 1/3 the tool diameter.

When selecting an RPM for non-ferrous metals, your range typically is between 30,000 – 50,000 RPM. A 6mm (1/4") single flute end mill should be around 30,000 rpm while a .6mm (.025") single flute end mill should be around 50,000 rpm. Therefore a 3mm (1/8") single flute tool should be around 40,000 rpm.

Feed rate is greatly affected by the RPM you choose and the type of machining. For example, a tool machining four surfaces (channel) compared to profile machining a pocket (one surface) would have substantially different feed rates. One guideline for the larger tools is, watch your spindle current as it should not exceed 60%. If the load exceeds 60%, reduce your feed rate. Another guideline is chips should be large, fluffy and warm. If this is the case, you are in the right ballpark.

When pocketing or contour machining we generally recommend using a single flute end mill for optimum feed rates and material removal. Because of the high rate of material removal, the surface finish sometimes is compromised. Therefore sometimes a finish cut is required with the single flute. If surface finish is critical and not speed, we recommend using a double fluted end mill at a lower RPM and feed rates.

One exception with Datron tooling is with our special line of high speed end mills. These tools are designed for fast contour milling. They leave a minor vibration mark on the edge, which can be easily removed with a finish pass. For example, the 2mm (.080") high speed end mill can machine 1/8" aluminum in one pass at 48,000 rpm with full coolant at 150 inches per minute. Because these tools do not fit the typical guidelines, we recommend you speaking with a Datron representative for speeds and feeds when using these specialty tools.