VERICUT®

CNC Machine Simulation Software

True Constant Chip Thickness Machining

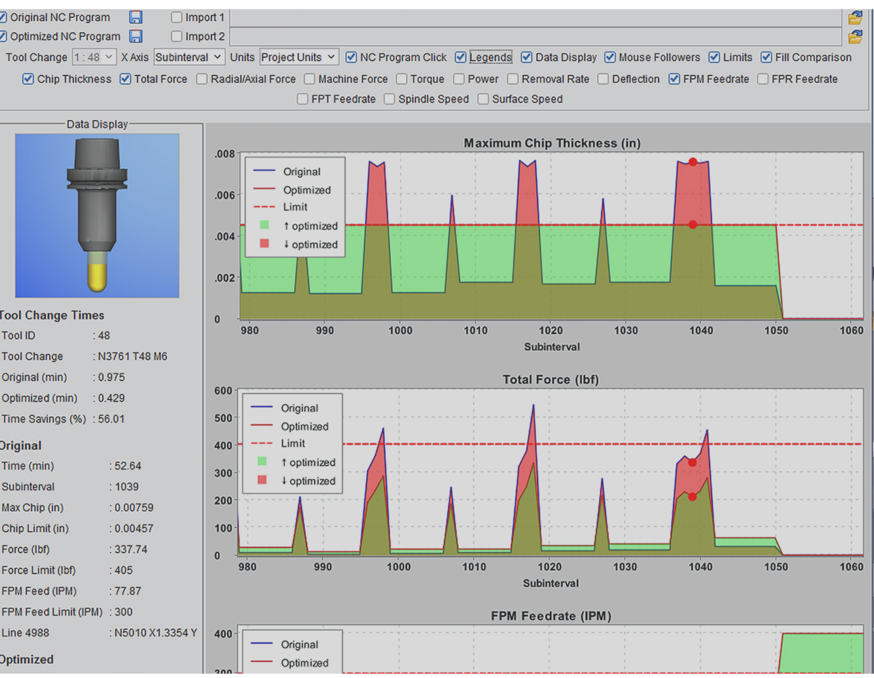
The New Standard of NC Program

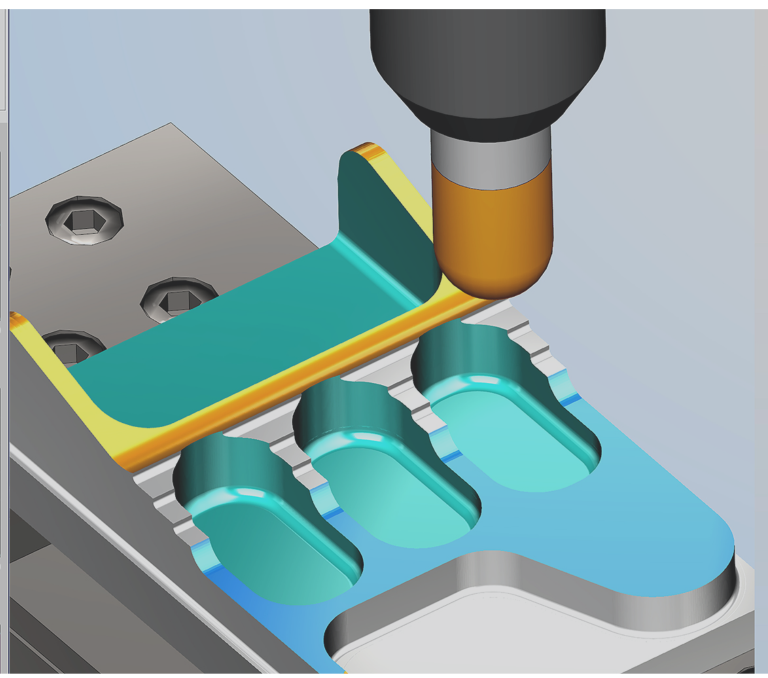
White Paper

Pete Haas

Product and Cutting Specialist

October 2021





CGTech

Introduction

There is a current crisis in the CNC machining (material removal) industry: cutting tools are greatly underutilized due to less-than-optimal NC programs that do not adjust feed rates as cutting conditions change. This results in costing nearly every metal cutting company time and money. Even worse, many manufacturing companies do not even realize this inefficient cutting is happening.

The good news is that this problem presents great opportunity for substantial time savings, increased metal removal rates, improved tool life, and higher quality parts, all of which add up to substantial cost savings. This paper will describe the issue in depth, how NC simulation software reveals this problem through new graphs and data, and the solution presented in advanced NC program optimization software.

Machine tool manufacturers, cutting tool manufacturers, C-Level and manufacturing managers looking to make productivity gains should take note: this is an opportunity to make substantial productivity and profitability gains with minimal risk. Almost any shop ready for the next leap in cutting tool and machine performance improvements would benefit from this new standard of NC programs.

Problem Statement

In milling and turning, the metal removal goal is maximum and constant chip thickness. The reality is that current NC programs do not meet this goal.

To use a quote from Sandvik Coromant:

> *Maximum chip thickness (hex) is the most important parameter for achieving a productive and reliable milling process. Effective cutting will only take place when maximum chip thickness is maintained at a value correctly matched to the material being cut and cutter in use.*
>
> - *A thin chip with a hex value that is too low is the most common cause of poor performance resulting in low productivity. This can negatively affect tool life and chip formation.*
> - *A chip thickness value that is too high will overload the cutting edge, which can lead to chipping and tool breakage.*[i]

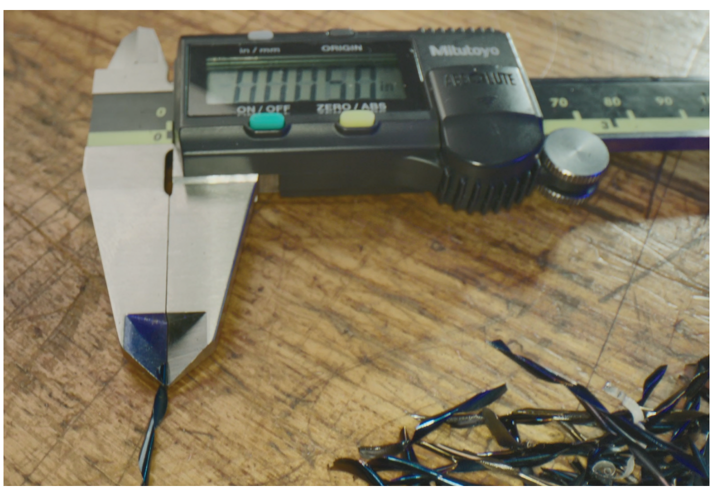


Figure 1 – Chips measured at .005 inch

Typical CAM-generated tool paths result in varying rates of material removal because they do not account nor adjust for changing cutting conditions caused by complex part geometries. With a typical NC program, the amount of material a tool might need to remove can be ten times greater or ten times less than the tool is designed to handle. This results in varying chip thickness, which results in less-than-optimal cutting.

Figure 2 – End mill and cut chips from 4340 steel

Maintaining a maximized and constant chip thickness allows the cutting tool to do as much work as it can, ensuring that it performs to its full engineered potential.

As stated by Tom Pyle, Product Development Manager at Harvey Performance Company:

> *The aim is to achieve a constant chip thickness by adjusting the feed rate when cutting at different RDOC. (Radial Depth of Cut, AKA [ae], AKA width of cut)* [ii]

The problem can be broken down into two parts: NC programs do not account for changes in cutting conditions, and underutilized tool capacity.

Problem 1 – No accounting for changing cutting conditions

The NC program output from any CAM system does not effectively account for the typical magnitude of changing cutting conditions, evidenced in chip thinning or variable tool engagement with material. These changing cutting conditions cause varying loads on the machine and cutting tool. This variability on tool load causes non-uniform wear and unpredictable tool life, which leads to catastrophic tool failure or even spindle damage.

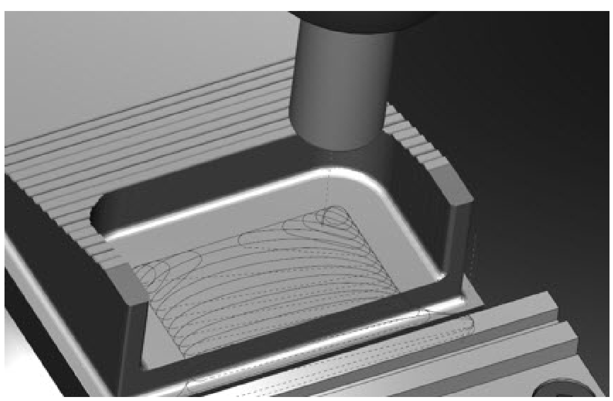


Figure 3 – Adaptive tool path in an open pocket

In Figure 3, even with the utilization of advanced tool paths, the cutting condition changes. With each pass through the pocket, the amount of material changes throughout the path.

NC programmers and shop personnel mistakenly believe that the NC programs sent to CNC machines will cut at the desired chip thickness throughout the whole program with a prescribed spindle speed RPM (S) and feed rate (F). In order to maintain a constant chip thickness throughout the process, the feed rate should adjust according to the cutting conditions. Maintaining a single feed rate and spindle speed results in the second part of the problem: underutilized cutting tools.

Problem 2 – Tool capacity is not fully utilized

Cutting tools have performance limits, which are cutting ranges where they perform best. They are also known as the upper capacity limit and lower capacity limit.

In Figure 4, the "Vc" represents speed and "Fz" represents feed per tooth. This is called the cutting tool capacity window or performance window. A cutting tool should successfully remove material if machining inside the performance window.

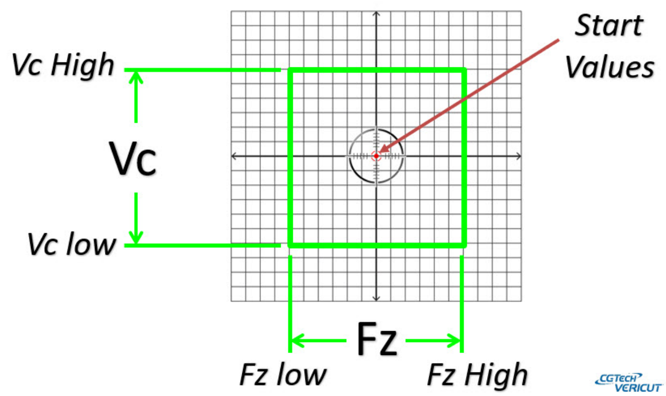


Figure 4 – Cutting tool limits (effective cutting ranges)

When cutting softer material, this performance window is relatively large since the material is forgiving to machine. Conversely, when cutting harder or more difficult to machine materials, the performance window is comparatively smaller because the cutting is less forgiving.

It is obvious when a tool is used beyond its capacity. It produces an audible scream, chips or breaks, but by the time a machinist receives this kind of feedback, it's too late to address. It is an indicator of damage to the tool, part, or machine.

It is less obvious when a cutting tool is being underutilized. To the machinist, the program seems to be doing its job as long as it results in acceptable part quality. The machinist may not know that the cycle time is running much longer than it should and wearing down the tool's life with premature wear from running too slowly.

In summary, this is the hidden reality of most NC programs today:

- Fixed Feed rate (F) causes chip thickness variation
- Poor cutting tool performance (excessive feeds, chip thickness, forces, deflection)
- Underutilized CNC machines & cutting tools
- Longer cycle times than necessary, causing shorter tool life and undue wear

NC Simulation Graphs Expose the Problems

How is an NC Programmer supposed to know if a cutting tool (milling solid tool, indexable insert, or turning insert) is being used to its full engineered potential? How can they verify that their tools will not suffer adverse effects like excessive feeds, incorrect chip thicknesses (too thin or too thick), excessive cutting forces, or tool deflection?

NC simulation software that provides detailed cutting condition graphs can provide visibility to the aforementioned issues.

Graphs provide a pictorial view of machining processes that have never been previously available.

The ability to have graphs in combination with the 3D digital twin graphics give an NC programmer powerful visual feedback to aid in making program improvements. A programmer can see a simulation of the cutting tool in the material along with the position in the NC program and graph showing the chip thickness, force and tool deflection at any point. Graphs can also provide insight into torque, power, material removal rate, and other useful data that an NC programmer can use to make intelligent machining decisions.

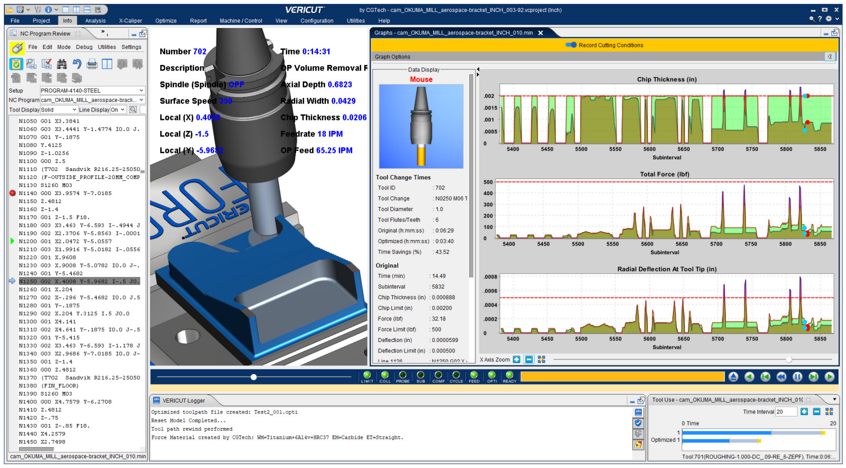


Figure 5 – NC Program, Digital Twin, Graphs for Chip Thickness, Total Force, Tool Deflection shown in VERICUT

Spikes in the graph may indicate an excessive cutting condition that may demand attention.

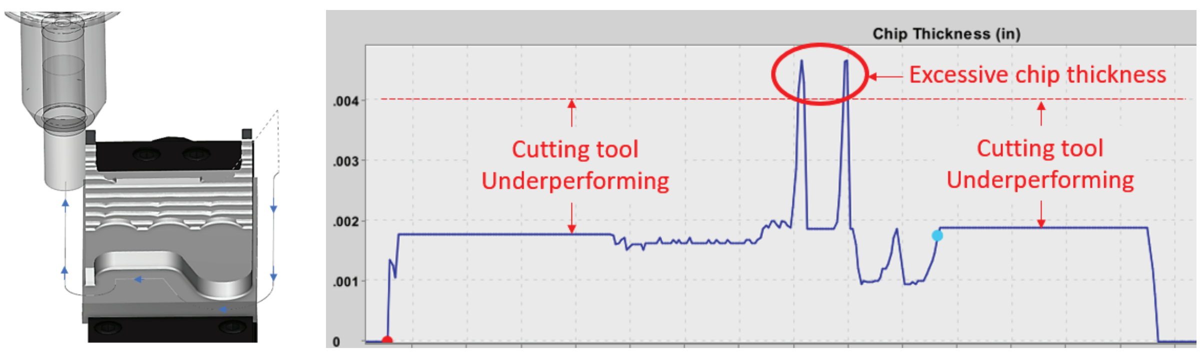


Figure 6 – Optimization Graph of profile cut

Graph data below or above the chip thickness target (indicated in Figure 6 by the dotted line) displays opportunities for improvement. A spike in the graph (chip thickness is over the limit) shows excesses where the feed rate should be reduced, while a valley (chip thickness is below the limit) means there is an opportunity to increase the feed rate.

Attempts at Constant Chip Thickness: What Has Been Tried

Adaptive Tool Paths

One strategy for improving cutting is adaptive milling, also known as Dynamic Milling, Profit Milling, VoluMill, and other names. These types of CAM operations attempt to produce continuous smooth tangent motions rather than traditional operations which have abrupt directional changes. This is where the term “adaptive milling” comes from, as the tool motion “adapts” to the profile geometry.

These CAM operation methods are designed to do two things: 1) make continuous, smooth paths and 2) make paths as parallel as possible to reduce cutting condition changes by controlling tool engagement angle and varying width of cut. These tool motion types are recommended as an improvement over a traditional tool path and do a better job with metal removal at the machine.

The issue with this type of tool path is that it is **not** true constant chip thickness machining. The output does not adjust the feed rates for changing cutting conditions while machining, such as entering and exiting each smoothing operation. Chip thinning still occurs on these adaptive type tool paths, leaving 15-20% of cycle time opportunity. Tool wear increases due to running slowly at entry and exit.

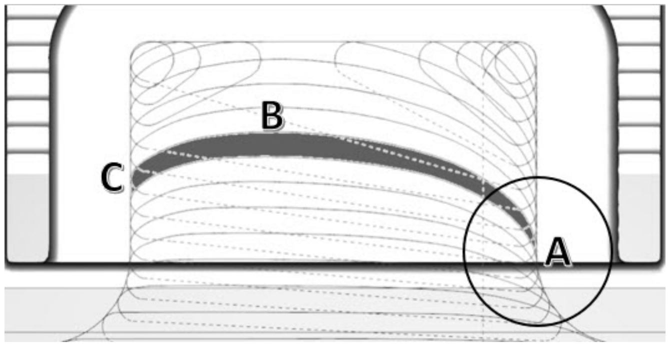


Figure 7 – Top-down view of open pocket adaptive machining path

Figure 7 is a view looking down on the part and cutter, showing an adaptive machining path for an open pocket. The lines inside the pocket are the centerline of the cutting motion. “A” indicates where the cutter is entering the adaptive cut. “B” is where the cut is fully engaged in the programmed width of cut. “C” is where the motion is exiting the cut then to return for another loop.

The dark crescent shape is the material being removed on a single pass. It starts out thin, gradually thickens, then thins out again. Looking at the NC program code for this adaptive operation, the cut is made using only one feed rate, with a fast traverse when not cutting material.

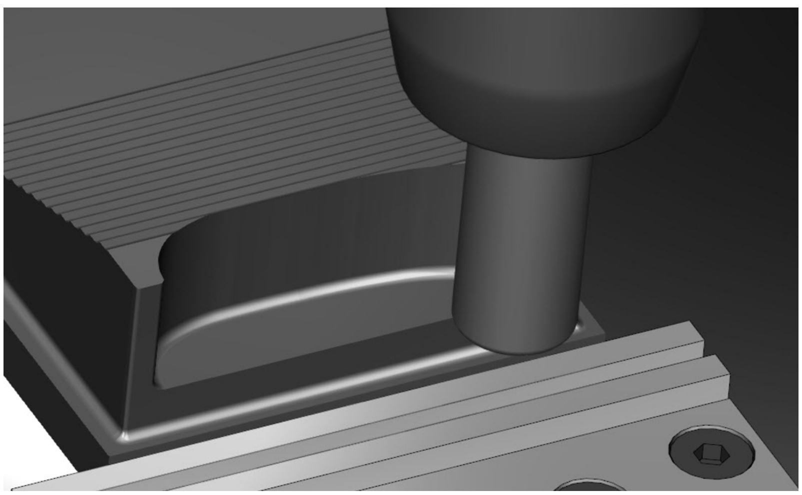


Figure 8 – Open pocket simulation view

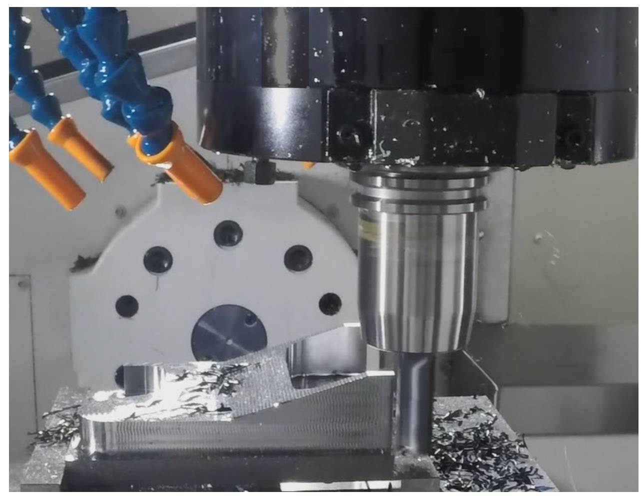


Figure 9 – Photo of the open pocket after machining

Chip Thinning Adjustments

Some CAM systems do have chip thinning adjustment settings, but these are mostly static settings and create a constant feed throughout the operation. This means that the feed rate is adjusted only once for all cutting motions. This is essentially similar to a machinist turning up the feed rate override.

Catalog Data, Chip Thinning Calculators, and Equations

Cutting tool manufacturers and all manufacturing professionals should be aware that catalog data, chip thinning calculators, or equations only provide the NC programmer with starting values for a certain set of criteria at a specific location in the NC Program.

$$f_z = \frac{h_{ex} \cdot DC}{2 \cdot \sqrt{(DC \cdot a_e - a_e^2)}}$$

Where: DC = cutting diameter

a_e = radial engagement

Figure 10 – Chip thinning formula

These start values are only accurate at this input condition. As the cutting conditions change, so should the feed rate. Otherwise, chip thickness will not be constant. Again, the NC Programmer who uses chip thinning calculations is not sending true constant chip thickness programs to the machine.

CAM Verification Systems

The verification methods included within CAM systems are inadequate for complete and accurate simulation and verification. A user should be able to catch material crashes, gouges, and watch the material being removed to make CAM operation improvements. But the CAM system's verification does not track data such as excessive cutting forces, deflection, or recognize improvement opportunities for faster feeds, which results in varying chip thicknesses. A more robust verification and optimization system is needed to identify these issues and make these changes.

The Solution: NC Program Optimization Software

The ideal solution comes through using an NC program optimization software that detects and resolves underperforming conditions, while simultaneously preventing excessive cutting conditions.

Solving Problem 1 – No accounting for changing cutting conditions

NC program optimization software analyzes the cutting tool interaction with the material through each block of the NC Program to determine whether to speed up or slow down feed rates along the programmed path. The software will break up and add blocks without changing toolpath trajectory to assign an ideal feed rate for each block. The result is true constant chip thickness machining (milling or turning). This level of optimization effectively acts like a machine operator, monitoring cutting behavior and constantly adjusting the machine's feed rate to eliminate varying chip thickness.

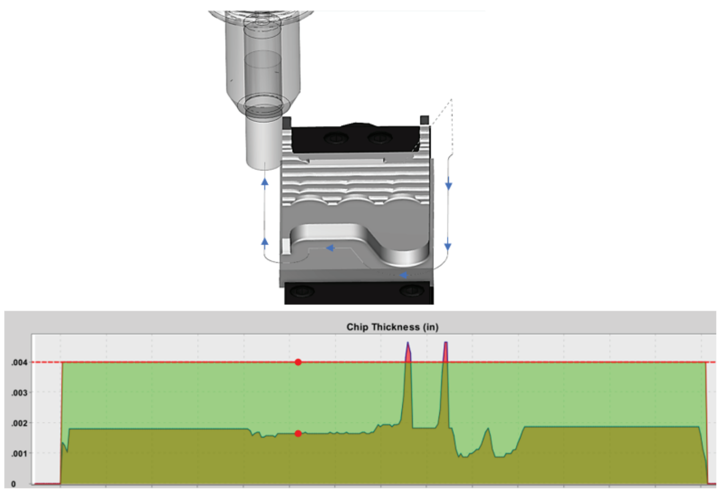


Figure 11 – VERICUT Force Optimization graph. Green areas indicate increased feedrates and red areas indicate where the feed was decreased

Figure 11 shows that the chip thickness is now maximized to the ideal value (.004" in this example) and held constant throughout the profile cut. As the cutting conditions change, the feed rate is changed to optimize the NC program.

Figure 12 shows the differences between the original NC program and the program optimized to the new standard – constant chip thickness cutting.

This is a fundamental change to the way NC programs have been and are currently created. NC programs typically hold the feed rate constant while the chip thickness varies, evident in the drastic shifts throughout dark green area in Figure 11. The light green area shows the optimization of the feed to generate the ideal maximized and constant chip thickness.

The new standard of NC program will have the chip thickness held constant by adjusting the feed rates as the cutting conditions change.

Figure 12 – Original NC program on the left, the new standard of NC program on the right

Solving Problem 2 – Tool capacity is not fully utilized

The new standard of NC program optimization is based on calculating physical forces acting on the cutter as it moves through material. This calculated force and chip thickness is used to adjust feed rates so the chip thickness will be as close to constant as possible without exceeding user-specified limits.

This ensures that the cutting tools are utilized to their full material removal potential without excessive wear or the potential of failure. Utilizing physics-based calculations to adjust for changes in cutting conditions results in superior metal removal rates, noticeably faster cycle times, and longer tool life.

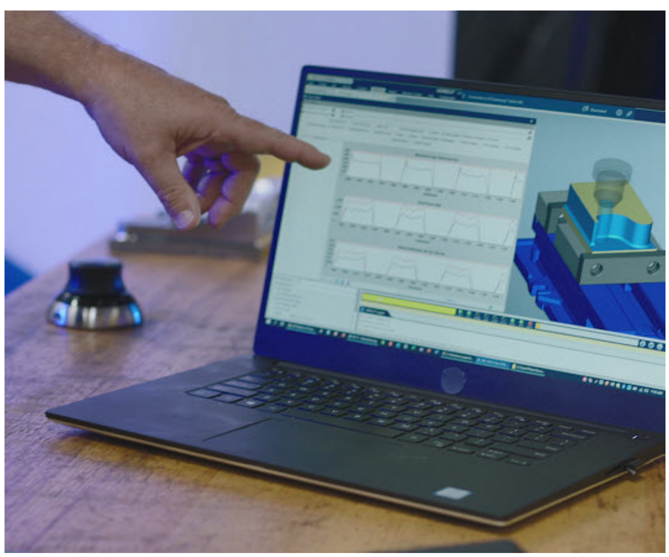


Figure 13 – Photo of VERICUT Force Optimization and graphs

This method of NC program optimization sets the new standard of excellence for NC programs used on CNC machines.

Conclusion

CAM systems' NC program feed rates are not the best they can be. NC program optimization software can improve new programs, as well as existing legacy NC programs without having to reprogram them.

Manufacturing companies that take advantage of NC program optimization software will lower manufacturing costs and improve profit margins significantly. Feed rate optimization will make a company more profitable and, for some, may be the key software tool that keeps them in business and outperforming the competition.

The results of optimization are:

- CNC machines are utilized to their full engineered potential
- Cutting tools are utilized to their ideal engineered performance
- Substantial cycle time savings, often greater than 50% on roughing tools*
- Tool life is greatly extended as tools run cooler to avoid unnecessary wear and tear
- Improve part surface finish
- Highly reduced machine and spindle wear and tear
- High value return with low risk

*Every machined part is different and time savings and tool life savings will vary. Machining improvements are a balance between tool life, speed, and surface finish.

What does this mean for a machine shop?

- Improve delivery and beat deadlines
- Increase capacity and throughput – get more done without purchasing an extra machine
- Win more jobs with an opportunity to reduce quote or bid price
- Better utilization of CNC machinist time by leaving machines running unattended
- Better utilize cutting tools with improved performance and longer tool life
- Better machine tool utilization with improved performance and operation
- Improved part-to-part and job-to-job consistency, regardless of the NC programmer
- Confidence when running unattended, lights-out machining

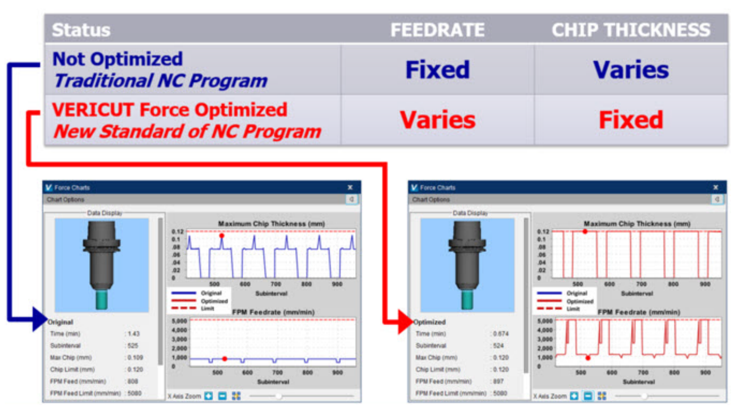


Figure 14 – Traditional NC program and results vs. Optimized NC program and results

This new solution recommends a fundamental shift in analyzing speeds and feeds. Until now the NC Program has a set spindle speed (S) and a set cutting feed rate (F). It is now possible to optimize programs for an optimal, fixed chip thickness and have a dynamic feedrate where the feedrate adjusts for cutting tool and material contact (see Figure 14).

The application and adoption of the new standard program produced by an optimization software must be put into practice for cutting tool manufacturers and part manufacturers to continue to compete in the CNC market.

Author

Pete Haas | VERICUT Product and Cutting Specialist for CGTech
pete.haas@cgtech.com

[i] https://www.sandvik.coromant.com/en-us/knowledge/milling/pages/entering-angle-and-chip-thickness.aspx, Maximum chip thickness in milling

[ii] Tom Pyle, How to Combat Chip Thinning, March 13, 2017/2, https://www.harveyperformance.com/in-the-loupe/combat-chip-thinning/