FANUC

WHITE PAPER

NOVEMBER 2015

High-Quality, Fast & Flexible 5-Axis Part-Centric Workflow

How to program FANUC 5-axis CNCs for the maximum part quality, machining speed, and flexibility

1 Introduction 5
2 Limitations of the Traditional Machine-Centric Workflow 7
2.1 Overview of the Workflow 7
2.2 Long Setups 8
2.3 Long Cycle Times, Large Programs, Poor Surface Finish 8
2.4 Inefficiency of Dedicated Machine Output 11
2.5 Inverse-Time Feedrates 11
2.6 Complex Expensive Post Processor 12
3 Benefits of the Modern Part-Centric Workflow 13
3.1 Overview of the Workflow 13
3.2 Quick Setup 14
3.3 Gouge-Free 5-axis Machining 15
3.4 Fast, Smooth Complex Curves 15
3.5 Feed-Per-Minute Feedrates 18
3.6 Programmable Axis Response 18
3.7 Flexible Modular G-code Programming 19
3.8 Vector Programming 21
3.9 Simple Inexpensive Post Processor 22
4 Improving Part Accuracy 23
4.1 Mechanical Accuracy 23
4.2 Pitch Error Compensation 23
4.3 Setting Accurate Kinematics 24
4.4 Basic Volumetric Compensation 25
4.5 Thermal Error Compensation 25
4.6 Advanced Volumetric Compensation 26
5 High-Speed Roughing 27
6 Summary 29

1 Introduction

Take a high-quality 5-axis machines with exactly the same CNC hardware and basic software. Process the same example part, once using the traditional 5-axis machine-centric workflow, and once using the contemporary 5-axis part-centric workflow. The same machine, with the same CNC, and he same amount of material is removed and the same amount of value is added.

However, you may notice some significant differences between the performance and quality metrics of the two processes as summarized in the table below.

Metric	Machine-centric workflow	Part-centric workflow
Setup time	2.5 hours	30 minutes
Cycle time	1 hour 20 minutes	40 minutes
Surface finish	Rough	Smooth
Portability	Can only run on a specific machine (production bottleneck)	Can run on any machine (with appropriate work envelope)
Post processor	Complex, expensive, customized for each machine tool	Simple, inexpensive, one for all machines

The performance improvements provided by the part-centric workflow are achieved using a handful of techniques and CNC features:

- Programming in part coordinates rather than machine specific coordinates
- Real-time spline interpolation with nano-precision
- High-gain tuning possible when using real-time spline interpolation
- Tool center point control (TCP)
- AI contour control

Though the comparison provided in the example above is a simultaneous 5-axis application, similar improvements can be expected anytime complex curves are machined with a 3-axis, 4-axis or "3+2" 5-axis machine. The results achievable somewhat depends on how much a machine has to be de-tuned to achieve an acceptable surface smoothness with the machine-centric workflow.

The aerospace, defense, and energy industries were early adopters of 5-axis machining because there was no other way to machine their large complex shapes to the required precision and surface finish quality. They pioneered the traditional machine-centric workflow because there was no alternative available at the time.

One of the challenges of being an early adopter is that emerging technologies improve significantly over time. Processes developed in the early stages may not be as efficient or as productive as those developed as the technology matures. When improved technologies are available, it is a challenge to update the established processes because of the limitations of regulatory controls, the shop is too busy to spend time introducing new practices, the fact that change brings risk and human resistance, or the simple lack of a process improvement culture.

Many manufacturers are still using the same primitive 5-axis machine-centric workflow technologies and processes that they developed 20 to 30 years ago. They may hear of competitors producing better quality and running at faster machining speeds, but assume that it because a particular machine and CNC brand combination is better than what they have in their own shop. In reality, it is likely that the other shop is simply using the modern part-centric workflow.

Today, an increasing number of manufacturers are moving to 5-axis machining for a variety of reasons. Some need 5-axis machining because consumer product shapes are increasingly becoming more complex. Others need the flexibility of 5-axis machining and combine multiple components into a more complex single component to reduce setup time, work-in-progress inventory and the overall shop cycle time, and to eliminate inter-process setup errors and assembly time.

These new adopters of 5-axis machining are used to the flexible part-centric 3-axis workflow where they simply program the part geometry – ignoring machine kinematics – and then use workpiece coordinate system offsets, tool length and radius geometry offsets run the program on any available machine.

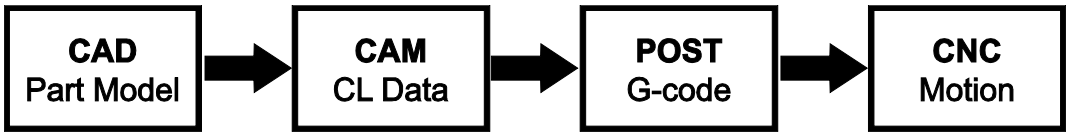
These shops expect the same convenience, flexibility and efficiency from their 5-axis machines, and therefore, they are more likely to adopt the 5-axis part-centric workflow. As a result, they are more efficient, effective and profitable businesses than the early adopters of 5-axis technology that are still using the machine-centric workflow.

Many of these efficient operations that use the 5-axis part-centric workflow are job shops and contract manufactures that serve the industries and OEM manufacturers that pioneered the traditional 5-axis machine-centric workflow. In today's global, competitive market place, these shops compete directly with the in-house machining operations that still use the less efficient 5-axis machine-centric workflow.

Quality 5-axis machines are expensive and it makes sense to maximize quality, productivity and efficiency by understanding the limitations of the traditional machine-centric workflow, the benefits of the part-centric workflow and how to systematically apply part quality improvement technologies to get the most return for the least investment.

2 Limitations of the Traditional Machine-Centric Workflow

2.1 Overview of the Workflow



The traditional 5-axis machine-centric workflow has four elements:

CAD – A design engineer uses a computer aided design (CAD) system to create a 3D model of a part. If the part includes complex curves, the curves are defined using splines.

CAM – A manufacturing engineer uses a computer aided manufacturing (CAM) system that takes the design engineer's model from the CAD system and translates it into a tool path. The tool path is a sequence of XYZ points on the surface of the part with an IJK vector that describes the required tool orientation at each point.

When the CAD model includes complex curves, they are most often converted to dot-to-dot linear segment approximations of the curve. Lots of points are required to ensure the curve is modeled.

The cutter location file (CL-File) output by the CAM system is completely part-centric and includes nothing that is specific to any particular machine.

Post Processor – A manufacturing engineer uses a post-processor to translate the part-centric CAM data into G-code – defining axis positions using the machine coordinate system for a specific machine. The post processor must have exact information about the kinematics of a particular machine tool including accurate pivot point positions, accurate geometry, length and radius of the tools that will be used to machine the part, and the exact location that the part will be placed within the machine coordinate system work envelope. Feedrates are hard coded on every line of the G-code program using the inverse time mode (G93).

CNC – The CNC interprets the G-code created by the post processor. It has no knowledge of the part geometry since the axis positions are in the machine coordinate system. The CNC and drive system attempts to faithfully reproduce the dot-to-dot linear segments output from the post processor – it simply connects the dots – and has no insight to the curve originally defined in the CAD system.

The machine-centric workflow is inefficient and inherently leads to inaccuracies.

2.2 Long Setups

In the machine-centric workflow, the axis positions in the part program use the machine coordinate system and assumes the specific part position and orientation, and the specific tool geometry specified in the post processor. There is no opportunity to use any offsets to correct for errors.

The part must be nudged and shimmed to the exact location and orientation that was assumed in the post processor – a problematic and time consuming task, especially with the large, irregular shaped parts found in many industries. Complex fixtures used to simplify setup still take time to changeover and accurately setup themselves. They also add cost and increase lead-time for new product introductions or process improvements.

All tooling must be carefully checked for wear and be preset to the length assumed in the post processor. This process does not impact cycle time, but it does increase overall costs and is somewhat inflexible.

2.3 Long Cycle Times, Large Programs, Poor Surface Finish

In an ideal world, the CAD, CAM and CNC would all be able to describe and interpret the same spline curve that was originally visualized by the design engineer. There are high-end CAD/CAM systems that can define NURBS (non-uniform rational b-splines) and CNCs that can accept NURBS paths in a proprietary format and interpolate the NURBS curve directly. Unfortunately, these system are very expensive and there are limited skilled resources to support them.

In practice, most CAM systems convert the splines defined in the CAD model to a series of point-to-point linear tool motions that approximate the curve using a specified tolerance (figure 1). When a tighter tolerance is specified, more points are generated by the CAM system, which results in larger part programs. In theory, a tighter tolerance should produce a more accurate profile when machined on the CNC. However, whatever the tolerance level specified, the technique still defines numerous sharp points rather than the smooth spline curve that was in the CAD model.

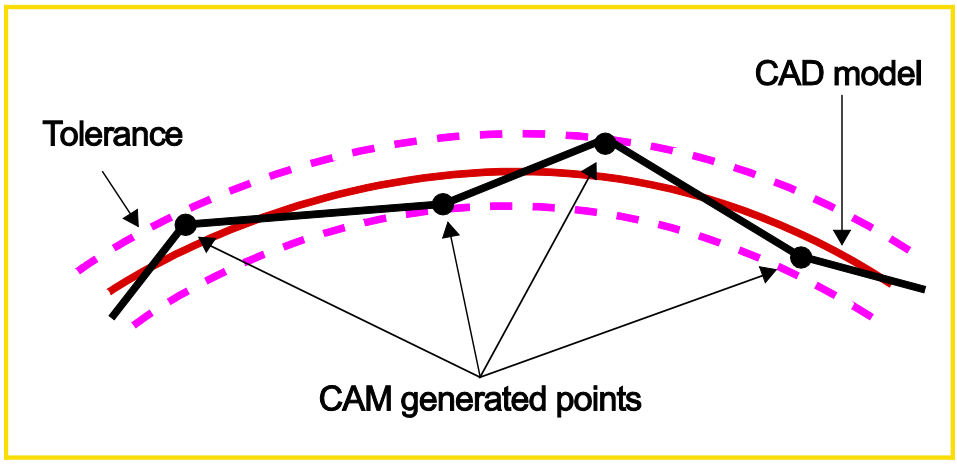


Figure 1: Point-to-point linear tool motions output by the CAM system to simulate a complex curve using a specified tolerance.

In the past, heavy, high-inertia machine tools and low-response CNC servo systems tended to smooth out the points. This produced a reasonable surface finish at the cost of an increased cycle time and reduced part accuracy.

Today's manufacturers are competing in a global market place that requires continuous process improvement. They purchase high-performance machine tools equipped with high-performance CNC and drive systems that are capable of exactly machining the CAM generated points – so the finished part is a sequence of points and not smooth.

When manufacturers using the old machine-centric workflow observe the unsmooth surfaces on a modern machine tool they fall back on old techniques to solve the problem. They detune the servo system and/or decrease the CAM tolerance to add even more points. Detuning the servo system increases the CNC servo system following error, reducing part accuracy and increasing cycle time. Adding more points does not solve the problem – there will still be points on the surface of the part, they are just less pronounced and closer together (figure 2). In fact it adds new problems as the technique significantly increases part program size and consumes CNC resources processing the numerous small blocks.

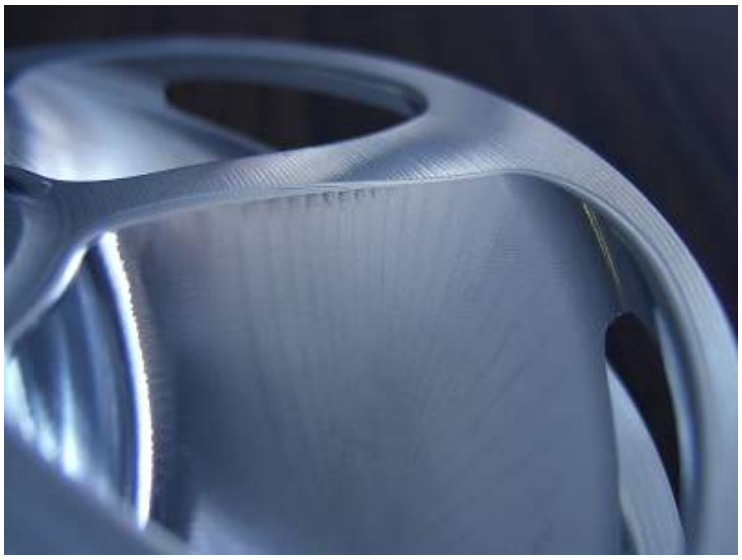


Figure 2: Stripes on the machined surface due to point-to-point linear segment programming and machine.vibrations.

The very nature of the dot-to-dot approximation of a curve limits the achievable feedrate of the part program. First, with a dot-to-dot approximation of a curve, the axes are continuously accelerating and then decelerating to a stop – never reaching their maximum achievable feedrates. Second, the jerky acceleration and deceleration cycles shock the machine creating vibrations that can be seen in the finished workpiece. To solve this problem the manufacturer may again detune machine servo performance – again, at the cost of an increased cycle time and reduced part accuracy – and/or the programmer must lower the effective feedrate.

The next problem is that scalloping will occur when moving between the points in the dot-to-dot approximation of the curve. It is important to consider that with the traditional machine-centric workflow, the CNC has no information about the desired work surface profile – the curve. The CNC is only provided the axis positions in the machine coordinate system that will, hopefully, produce the correct work surface profile. Therefore, the CNC cannot perform any compensation for kinematic errors.

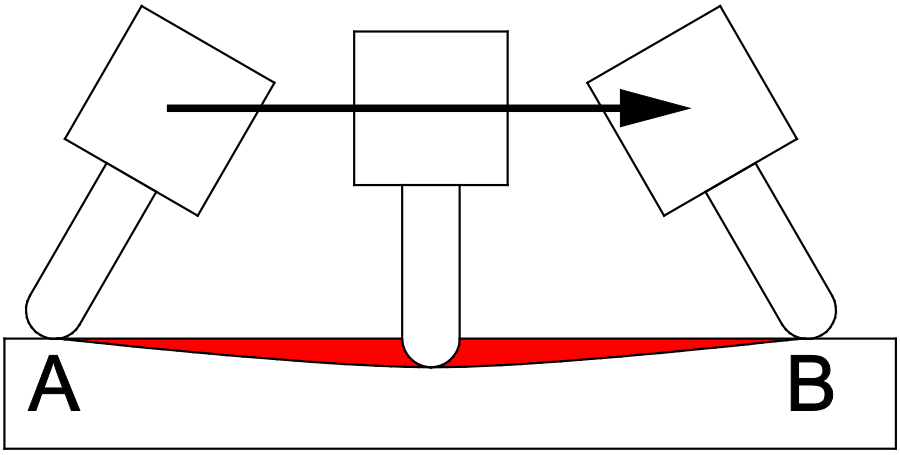


Figure 3: Surface gouging as the B-axis rotates between -30° and +30° because the CNC does not have a definition of the desired cutter path.

Consider a tool move between point A and point B. This could be an actual linear move or part of a dot-to-dot curve definition – the CNC does not know the difference. In the example, the program includes a movement for one of the linear axes (X) and a movement for a rotary axes (B), from about -30° to +30°. The tool-tip trajectory gouges the material as the rotatory axis moves through its arc (figure 3).

This is an extreme example to illustrate the problem. Some CAM systems may detect the gouging for an actual linear surface and flood the area with smaller segment moves. Again, this simply reduces the size of the gouging, which improves accuracy but still leaves lines on the workpiece surface. We now have more acc/dec cycles that will introduce more machine vibration and surface finish problems. In the case of the dot-to-dot approximation of a complex curve, as long as the calculated gouging is within the tolerance band, the CAM system is unlikely to do anything to compensate for the gouging and the problem will be visible in the surface finish of the part (figure 4).

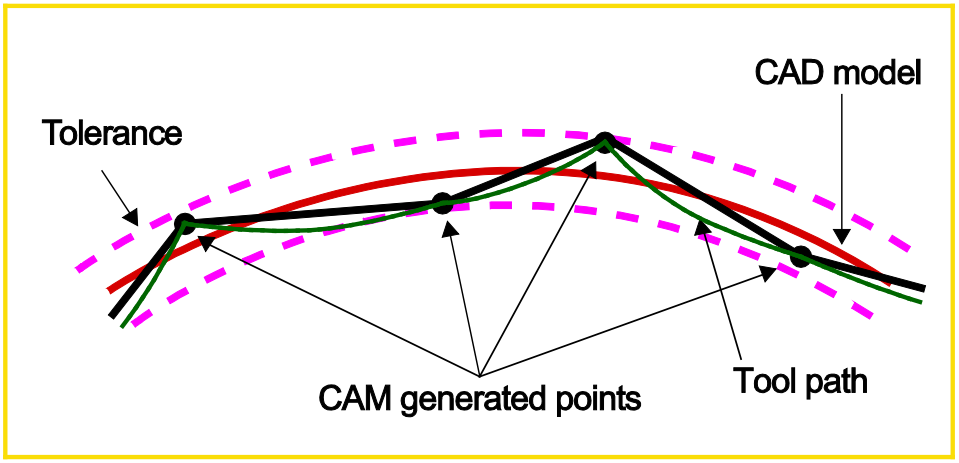


Figure 4: Gouging on a complex curve approximated by point-to-point linear segments – within the tolerance band but not accurate or a smooth finish.

Though the example above describes what happens in the case of machining with the end or tip of the tool, a similar condition occurs when machining with the side or diameter of a tool.

2.4 Inefficiency of Dedicated Machine Output

The G-code program created by the post process using the machine-centric workflow requires that accurate process information be stored in the CAM-post processor system including:

- kinematics of each specific machine with pivot points
- setup of the workpiece within the machine coordinate system
- tool geometry (type, length, radius)

If any of the factors change, a new part program must be generated.

This means a custom part program must be created for every machine that runs the part, since the kinematic details will not be identical for any two machines, even if the machine is from the same manufacturer, is the same machine model and has an identical CNC vintage.

The exact measured kinematics are required, not just the machine tools builder's nominal values. These will change over time, even for a specific machine, due to mechanical machine wear or accidental collisions.

As a result, manufacturing operations have a significant part program management issue and there are significant overhead costs to debug each custom version of the program. Therefore, production bottlenecks and idle machines are likely.

Another challenge is that because the part program uses machine coordinates, there is no correlation with the part model or blue print to aid the operator. Any editing to solve machining problems is very difficult, if not impossible.

2.5 Inverse-Time Feedrates

Traditional 5-axis machine-centric workflow programs use inverse-time feedrate mode (G93) to ensure that all axes can arrive at the destination end-point at the same time. The F-word specifies the time (1 over the F-word minutes) that the axes take to move between the current position and the destination end-point.

To calculate the correct feed rate the CAM system must know the exact kinematic distances on the machine. Every program block must include an F-word to set the time for a particular combination of linear and rotary axis moves.

Again, this makes the program machine-dependent and makes it very difficult for the operator to optimize feedrates during machining. In effect, you are hard coding the cycle time of the whole part program one block at a time. There is no opportunity for the CNC to use real-time, dynamic optimization techniques to improve cycle time, so process improvement is limited.

Since the F-word is in time, it has no connection with the cutting tool feedrate or the cutting feed recommendations from the tooling manufacturer.

2.6 Complex Expensive Post Processor

The algorithms that translate the XYZ tool tip positions and IJK tool vectors into specific axis positions by taking into account the machine coordinate system, a specific machine's kinematics and considering the workpiece setup and tool geometry, are incredibly complex. Just consider the different translations required for a trundle-table type 5-axis machine versus an articulated-head type 5-axis machine and realize there are many variations within those base types (figure 5).

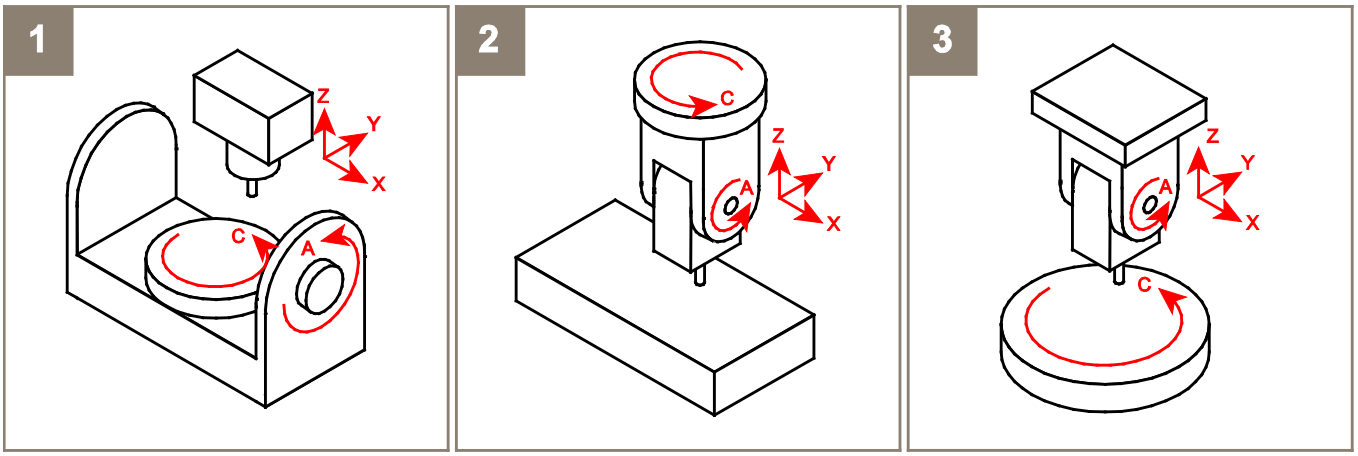


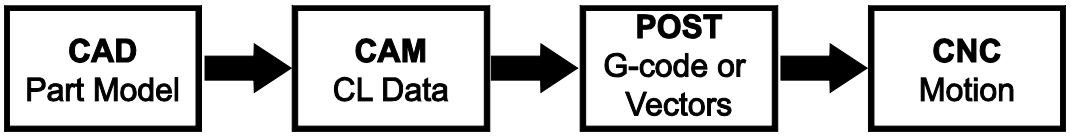
Figure 5: Variety of 5-axis machine kinematics: 1. trundle table type 2. articulated-head type 3. Hybrid type.

Using a single machine type for every part, from simple to complex and from small to large, is not very efficient or cost effective – so it is likely the post development must be repeated for several combinations of machine kinematics and CNC vintages. It is very hard to improve processes by tweaking the post processor without creating bugs in the base algorithms.

Considering all the CAM-post processor functionality described above, it is no wonder that the post processor development for each machine and CNC combination is likely to be expensive and complicated to produce and maintain.

3 Benefits of the Modern Part-Centric Workflow

3.1 Overview of the Workflow



At a first glance, there appears to be very little difference between the part-centric workflow and the machine-centric workflow. Indeed the first two elements are identical. The 5-axis part-centric workflow also has four elements:

CAD – A design engineer uses a computer aided design (CAD) system to create a 3D model of a part. If the part includes complex curves, the curves are defined using splines.

CAM – A manufacturing engineer uses a computer aided manufacturing (CAM) system that takes the design engineer's model from the CAD system and translates it into a tool path. The tool path is a sequence of XYZ points on the surface of the part with an IJK vector that describes the required tool orientation at each point.

When the CAD model includes complex curves, they are most often converted to dot-to-dot linear segment approximations of the curve. However, the part-centric workflow requires much fewer points to accurately model the curve.

The cutter location file (CL-File) output by the CAM system is still completely part-centric and includes nothing that is specific to any particular machine.

Post Processor – A manufacturing engineer uses a post-processor to translate the part-centric CAM data into G-code – defining axis positions using the workpiece coordinate system.

The part-centric workflow offers a choice of part program formats:

Conventional G-code – the program will include the positions of the axes in the workpiece coordinate system. This requires some translation, but the XYZ values in the CAM system CL-File only require relatively simple translations and the IJK values must be translated to provide the correct orientation of the tool using the rotary axes.

Vector Programming – the G-code program generated contains the same XYZ tool tip positions and IJK tool vectors as the CAM CL-File, just formatted into G-code.

Complex curves are still translated as dot-to-dot linear segments that approximate the curve, but much fewer points are required. Feedrates are specified, usually only at tool change points, in feed-per-minute (G94) mode. This is the actual cutting tool feedrate recommended by the tooling manufacturer for the application.

CNC – The CNC interprets the G-code generated by the post processor. The CNC translates the workpiece coordinates into machine coordinates for the specific machine being used, locally compensating for the machines kinematics and the actual workpiece location and orientation. If commanded, the CNC interprets the dot-to-dot linear segments as a definition of the original spline defined in the CAD system, machining the profile using real-time spline interpolation – converting the dots back to a curve.

The essential difference of the 5-axis part-centric workflow is that the CAD, CAM and post processor are all defining what the finished part should look like rather than the positions of the axes on a particular machine. The post processor is very simple and inexpensive compared to the one used in the machine-centric workflow. The CNC automatically compensates for the specifics of the machine-specific kinematics, the workpiece location and orientation and the actual tool geometry. All the machine and setup information stays at the individual machines and does not have to be constantly updated on the CAM system.

Since the CNC receives a definition of what the finished part should look like and has the performance to make adjustments and compensations in real-time, the finished part will be a closer representation of the original CAD model, with better surface finish and produced in less time.

3.2 Quick Setup

The part program generated by the CAM-post processor system uses the workpiece coordinate system referenced to a theoretical workpiece origin and part orientation. The 5-axis workpiece setting error correction feature compensates for the actual part location and orientation on the table in a similar fashion to the way that standard workpiece offsets (G54) does for common 3-axis machining.

Workpiece Setting Error Correction (G54.4) compensates for both the position (X-Y-Z) on the table and any orientation errors (α-β-γ) (figure 6). Since a spindle probe can easily locate the part on the table and determine its actual position and orientation, setup is limited to securing the workpiece and running the high-speed probing routines. Positioning and shimming of the part to an exact location are eliminated, a time consuming processes, especially with large and complex shaped workpieces.

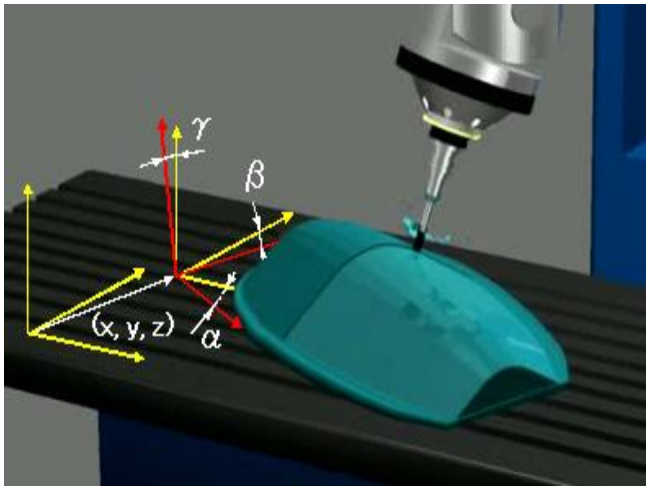


Figure 6. The CNC compensates for position and orientation of the part on the table.

Programming the part rather than machine and using workpiece setting error-correction significantly

reduces labor costs since operators are spending time shimming a part on the work table to match the planned setup. With large parts, it also allows for a safer work environment. Programming the part rather than the machine also makes the CAM system simpler and less expensive to maintain.

3.3 Gouge-Free 5-axis Machining

The primary function of Tool Center Point Control (TCP) (G43.4) is to translate the tool-tip path defined in the part program into the position and orientations for the machine's linear and rotary axes. It also allows the cutting tool feedrate to be specified in feed-per-minute mode (G94). Running at the tooling manufactures recommended cutting rate improves tool life. However, the CNC will automatically adjust feedrates to stay within the maximum performance of a limiting axis.

Tool center point control solves the problem of local tool gouging. Since the CNC knows the profile of the workpiece surface, it can coordinate all axis to keep the tool tip exactly in contact with the surface when moving between points. So when the program includes an X-axis move between A and B and a B-axis orientation change from -30° and +30°, the CNC automatically coordinates the Z-axis to keep the tool tip on the surface path (figure 7). This eliminates the gouging characteristic of common machine-centric workflow using inverse-time feedrates (G93).

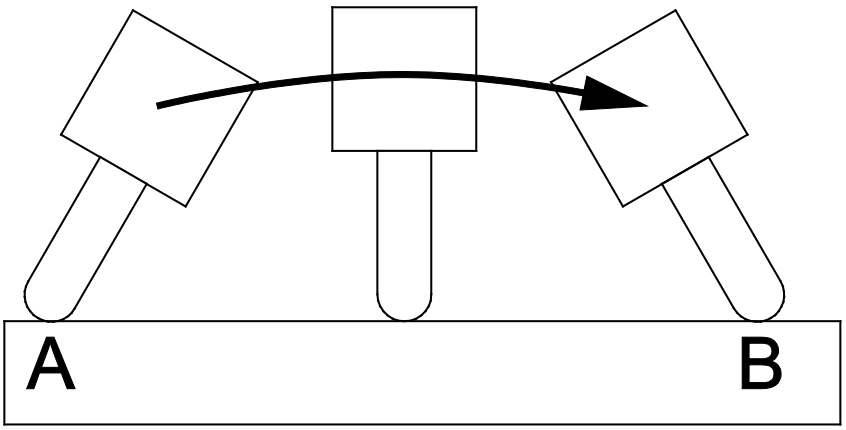


Figure 7. No surface gouging as the B-axis rotates between -30° and +30° because the CNC knows the part geometry.

Tool posture control performs a similar function when machining with the diameter rather than the tip of the tool.

Tool center point control and tool posture control improve part surface quality by eliminating scalloping and allow tool tip feedrates to be programmed in a single block, simplifying and shrinking the program by eliminating inverse time feedrates that are required in every block.

With the CNC taking care of potential gouging situations, the CAM-post processor system can be simplified and part programs are smaller.

3.4 Fast, Smooth Complex Curves

Design engineers define complex curves for sculptured surfaces using splines. The CAM system approximates the complex curves using small linear segments with a specified tolerance. The modern CNC will faithfully reproduce those small segments generating witness lines on the part.

Because of the constant acceleration and deceleration of the axis and the fact that the abrupt changes cause machine shock and vibration, feedrates are limited.

Some CNCs provide a NURBS spline interpolation mode and can machine the exact curve defined in the CAD model. However, the NURBS G-code format is not standard between CNC manufacturers and few CAM systems support all the variations.

When real-time spline interpolation is active, the CNC uses the dot-to-dot linear segments generated by the CAM system as a model for the curve that is actually required. The CNC looks ahead in the program and internally recreates a model of the spline that was originally defined in the CAD system (figure 8). The resultant smooth spline can be machined much faster than the dot-to-dot linear segments because the axes are not accelerating and decelerating as much. They are also not stopping and reversing direction as often. Real-time spline interpolation also minimizes the vibration and mechanical shock and vibration associated with dot-to-dot linear segments – extending the machine and tool life. The tool path is more likely to reach the programmed feedrate recommended by the tooling manufacturer, which is a positive for both productivity and tool life.

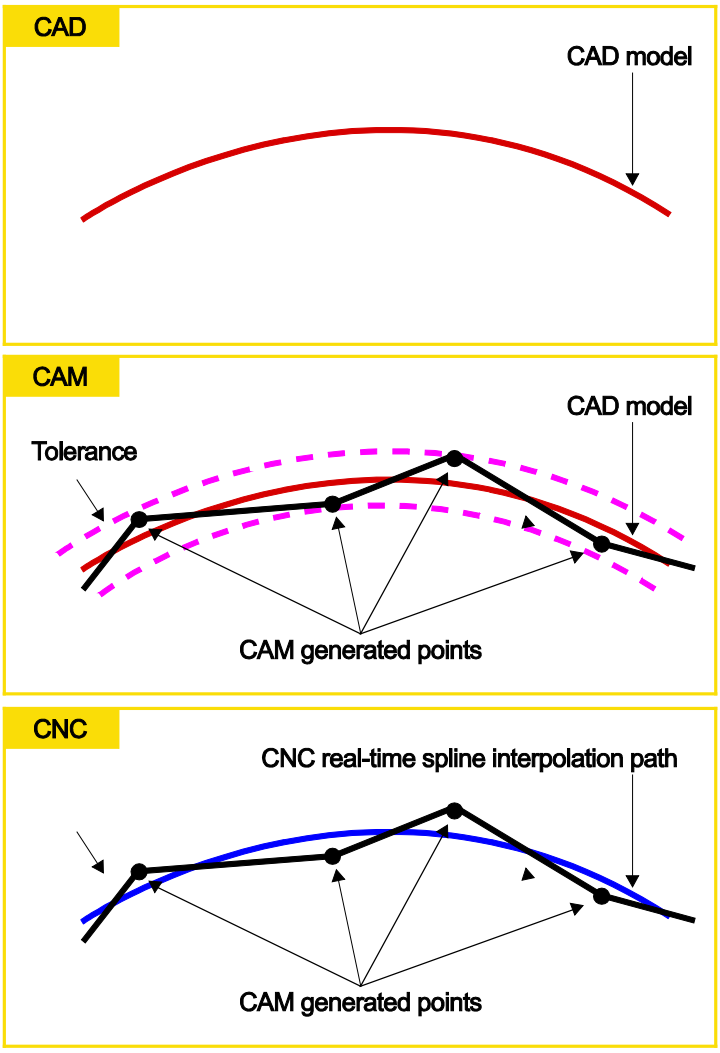


Figure 8. CNC real-time spline interpolation produces a curve closer to the CAM model than linear segments.

Translating the point-to-point linear segments using real-time spline interpolation allows the machine and servo system to be tuned to their maximum capabilities, reducing servo following error and improving machining accuracy. Eliminating the linear segment points also eliminates the witness marks on the workpiece surface. Scalloping is also eliminated by tool center point and posture control (figure 9).

The actual CNC curve machined will be much closer to the original CAD model than the linear segments defined in the CAM system while providing higher accuracy, a smoother workpiece finish, and much higher machining speeds.

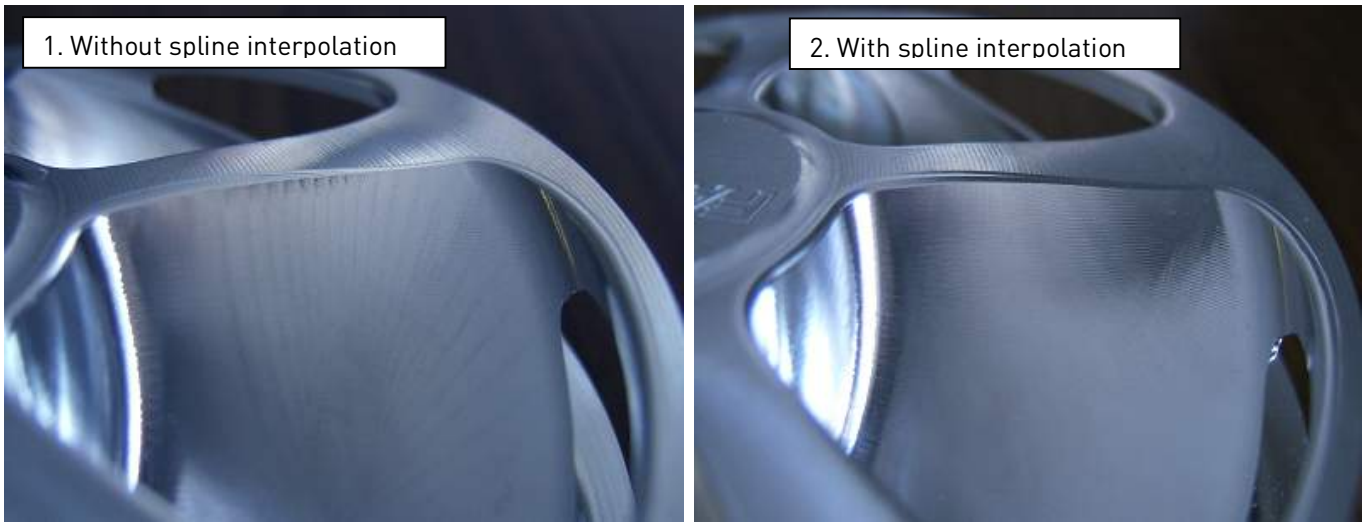


Figure 9: 1. Witness marks caused by linear segments and machine vibration 2. Smooth surface created by converting linear segments to spline curve and eliminating machine vibration

Activating real-time spline interpolation is a single line G-code, so it is easy to modify existing programs to take advantage of the feature. A single part program block activates real-time spline interpolation mode, and a second block deactivates it. All the part program blocks between the activation and deactivation blocks are standard line segments. Because we are now machining smooth curves rather than point-to-point linear segments, the CNC can be highly tuned, increasing the achievable feedrate, reducing the cycle time and improving part accuracy.

The G5.1 Q3 command activates real-time spline interpolation for each axis specified in the block (figure 10). Axes are specified with their address letter (X/Y/Z/A/B) and a zero value. No motion is commanded by the activation block. Activating real-time spline interpolation also activates AI contour control, providing automatic velocity control to further smooth impacts on the machine's mechanical systems. The G5.1 Q0 command deactivates real-time spline interpolation and AI machining contour control.

```
G5.1 Q3 [X0] [Y0] [Z0] [A0] [B0]; (activate real-time spline interpolation)
..
..  } Linear segment blocks
..
G5.1 Q0; (deactivate real-time spline interpolation)
```

Figure 10: Real-time spline interpolation mode activation and deactivation requires only a small modification to the G-code program.

For new programs, it makes sense to minimize the number of points generated for a spline curve by using a wider CAM tolerance than is traditionally used with the machine-centric workflow – using the optimal number of points will make programs shorter without sacrificing precision.

It is important to note that, if a machine servo system has been de-tuned to try and eliminate dot-to-dot witness marks, the servo system must be re-turned to take full advantage of the faster machining speeds that real-time spline interpolation function can provide.

3.5 Feed-Per-Minute Feedrates

Part-centric workflow programs allows the cutting tool feedrate to be specified in feed-per-minute mode (G94). Feedrates are easy to calculate and apply using the tooling manufactures recommended cutting rate. Using the tooling recommended cutting rate improves tool life.

The feedrate only needs to be programmed based on the application and according to the tooling manufacturer's application guidelines. This means that an F-word is only required at a tool change point or for specific applications, just like standard 3-axis machining.

This makes it easy for the operator to optimize feedrates during machining. The programmer can use optimistic feedrates and rely on the CNC to use real-time, dynamic optimization techniques to improve cycle time and maintain part accuracy.

3.6 Programmable Axis Response

Most machine tools from the factory have a very basic servo tuning, with the goal of providing reasonable performance on a wide range of part geometries and material types. Many 5-axis machines are used in industries such as aerospace structures or jet engine impellors, energy turbine blades or medical devices. These machines are typically used for a very specific geometry and a handful of material types.

Some machines are so dedicated to a particular part or material type that they can be optimized for that application, adjusting the acc/dec rates, jerk parameters and maximum cutting feedrate to produce superior processing speeds, part accuracy and surface finish quality.

The Machining Condition Selection function provides up to ten sets of optimized servo settings allowing for optimization based on the application (figure 11).

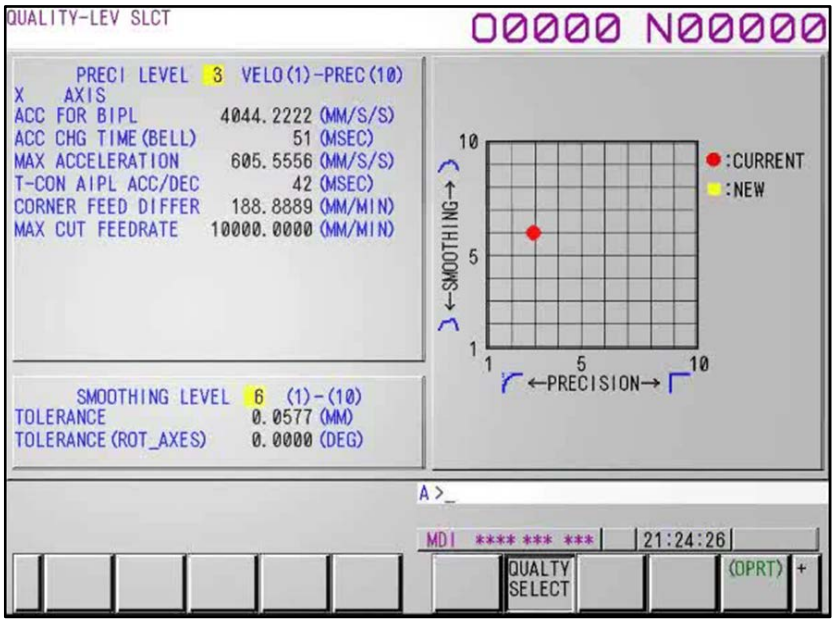


Figure 11: The machining condition selection function allows optimized axis performance based on machining application.

For example, when rough machining, speed is more important than accuracy.

When profiling, accuracy is more important than speed. And when making finishing passes, surface finish is more important than speed. Three difference sets of servo tuning parameters can be optimized and the program can easily switch between them in the part program.

Similarly, more aggressive acceleration and deceleration may be possible when cutting soft materials like aluminum, whereas some of the very hard exotic materials used in aerospace manufacturing and other industries may require a more conservative approach. Different sets of parameters may be activated based on the materials being machined.

The Machining Condition Selection function allows the machine to be optimized for different applications and eliminates the need for dedicating machines to a specific task. And because the parameters can be activated in the part program, it eliminates any drastic reconfiguration of the setup to switch between applications. This provides more manufacturing flexibility and minimizes bottlenecks.

3.7 Flexible Modular G-code Programming

Industries that use 5-axis machining extensively typically have machines from several machine tool builders, several models and vintages from the same machine tool builder, as well as several CNC vintages. Some of the variation is dependent on the application and some is based simply on when the machines were purchased. Some machines may be older machines that were retrofitted with a new CNC.

Over time, the CNC features for 5-axis machining improve and the detailed G-code to activate the features or certain feature options may change. Modular G-code Programming is a way to keep all the machine and CNC-specific G-code embedded in the CNC and let the CAM system activate those features at a high level. This is a common concept in computer programming. Modular G-code programming keeps the post processor as simple and inexpensive as possible and makes programs and the associated approval process flexible and machine independent.

Modular programming uses subprograms or Custom Macros to activate features, both of which are common features on 5-axis CNCs. Custom Macros have the advantages in that you can pass them parameters and include decision making structures for maximum flexibility.

The subprograms or macros can be stored in the CNC library folder to avoid unauthorized editing or accidental deletion.

Using modular programming, the main program begins with a call to a macro to measure the workpiece setting error correction values, typically using a spindle probe. Next, a call is made to a macro to activate all the 5-axis features to be used (figure 12). The main body of the program is simply the geometry of the part in part coordinates. Before the end of the program or before a tool change, the 5-axis features are all deactivated with a call to another macro.

```
O0001 (SAMPLE MODULAR PART PROGRAM
;
G65 P9004 (MEASURE WSEC VALUES);
;
G65 P9006 (ACTIVATE FEATURES);
;
(PART GEOMETRY STARTS HERE);
G00 X0.0 Y0.0;
Z1.0;
G01 Z-1.0 F100.;
X10;
Y10.;
X0;
Y0;
G00 Z1.0;

..
..
(PART GEOMETRY ENDS HERE);
;
G65 P9008 (DEACTIVATE FEATURES);
;
M30;
```

```
O9006 (ACTIVATION MACRO);
G54.4 P0;
G80 G40 G69;
G90 G49 G70 G94 G17 G64;
G54;
;
G54.4 P1; (WSEC)
G5.1 Q1 R1.;    (AiCC + MCSF)
G43.4 H1 P3.; (HSSTCP)
G5.1 Q6.; (FAIRING)
;
M99;
```

```
O9008 (DEACTIVATION MACRO)
G40;
G5.1 Q4;
G49;
G5.1 Q0.;
;
G91 G28 Z0;
G90;
M99;
```

Figure 12: Modular G-code programming example with calls to a 5-axis activation macro (O9006) and deactivation macro (O9008).

A side benefit of modular programming is that the G-code inside the macros is tested and approved once. The only new code to be tested for a new application is the actual part geometry. This simplifies verifying new programs and should simplify any compliance requirements. If a CNC feature is added or changed in the activation macro, only the activation macro has to be re-tested. The changes should be automatically proven for all new and existing part programs. The same is true for a new vintage of CNC with a potentially different feature sets. The macros simply activate and deactivate the best feature set to machine the part, but the base program from the CAM system is identical for all machines and CNC vintages. When multiple tools are used in an application, macro parameters can be used to specify the tool type and geometry. Custom Macro code can search the tool library for a tool that matches the specification and activate it.

3.8 Vector Programming

How well a 5-axis machine reproduces the original CAD design of a part is often limited by how accurately the CAM-Post system supports the kinematic of a specific machine tool. The post processor must convert the binary XYZ tool tip position and IJK tool orientation vector into the positions of the axes available on a particular machine. This is a complex translation and any small error in the kinematic model in the CAM-Post introduces errors when the program is executed on the machine.

In addition, the inclusion of specific axis letters and the assumed kinematics automatically limits flexibility. Different machine kinematics support different rotary axis configurations and their associated axis letter addresses (A, B, and C) (figure 5).

Vector programming sidesteps many of these problems by allowing the CNC to execute the exact XYZ tool center point and IJK tool orientation vectors generated by the CAM system (figure 13). The post processor simply has to convert the internal binary format to an ASCII text format. This significantly reduces its complexity and therefore its development and maintenance costs. The simple translation also means that post processing time is reduced significantly.

When combined with the modular programming technique, the same post processor can be used for all 5-axis machines, even though they may have very different kinematics and CNC vintage. All development, debugging and optimization of the post processor are automatically applied to all machines, multiplying the productivity gains.

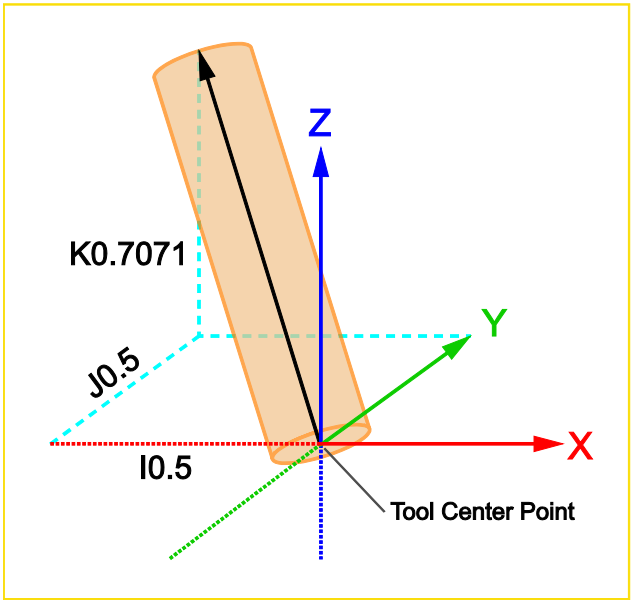


Figure 13: XYZ tool center point and IJK tool orientation vectors.

When using vector programming, CNC is totally responsible for compensating for a specific machine's kinematics. The machine kinematic values can be measured, calibrated and updated automatically using a spindle probe right at the machine, eliminating the need to continuously update the CAM-post processor system and re-generate and re-test previously proven programs. This completely separates the programming function from the machine maintenance functions. Programmers are responsible for defining the part geometry and process. Maintenance and production engineers are responsible for the condition and accuracy of the machine tool.

Using a high-precision minimum increment for the program and nanometer precision for all calculations inside the CNC ensures the translation performed produces the optimum part fidelity during machining.

Any modification required at the machine related to the workpiece geometry will be in the vector format, and the altered program can be uploaded back to the server and still be valid for all machines. Quality issues are reduced because the simple post processor can be thoroughly debugged.

The efficiency and flexibility realized from the combination of vector and modular programming provides a competitive advantage. OEM manufacturers and contract job shops realize new levels of scheduling flexibility and increased machine capacity – any program can run on any machine that can handle the workpiece size and weight.

OEM operations are immediately more competitive with outsourcing pressures. Contract job shops can market their flexibility and capacity to win new orders, building their customers' confidence in their manufacturing technology and ability to deliver on time.

Procuring a new machine is greatly simplified. A proven post processor is already available. All that is required is to run a test part on the new machine during trials to ensure all the correct options are available and that they are configured correctly. Modular programming allows the adoption of newer advanced features to deliver more productivity and accuracy gains, while the actual part program remains the same.

3.9 Simple Inexpensive Post Processor

When using the part-centric workflow, the algorithms that translate the XYZ tool tip positions and IJK tool vectors into specific axis positions do not have to take into account the machine coordinate system, the workpiece or the tool geometry, so they are much simpler than required by the machine-centric workflow.

Using modular programming and particularly when taking advantage of vector programming, the machine specific functions and kinematics are maintained at the machine and compensated for by the CNC in real-time. This means any changes to the actual machine do not have to be updated in the CAM-post processor system. Programmers are left to program the part and production and maintenance are responsible for the condition of the machining process. The post processor simply has to reformat the binary CL-File XYZ/IJK data into formatted G-code – the XYZ and IJK values are identical.

A single post processor can be used for a wide variety of machine tools and CNC combinations. The simple post processor is easy to create and maintain. Process improvements can be made at each machine without affecting the post processor.

Considering the limited post processor functionality required by the part-centric workflow using modular programming and vector programming, the post process is likely to be inexpensive and much easier to produce and maintain.

4 Improving Part Accuracy

The part-centric workflow discussed previously improved part accuracy using real-time spline interpolation to reproduce the CAD design curve and by allowing the CNC drives to be optimized to reduce servo following error. Real-time spline interpolation reduces part accuracy issues introduced by the machine-centric workflow.

This section discusses CNC features that improve part accuracy by compensating for machine mechanical issues.

It is important to consider machine accuracy in a specific sequence to maximize effectiveness and minimize effort and cost:

1. Mechanical accuracy repeatability
2. Pitch error compensation
3. Accurate measured kinematics and tool lengths
4. Basic volumetric compensation (e.g. straightness, squareness and beam sag)
5. Thermal compensation
6. Advanced volumetric compensation

It makes sense to achieve the largest improvement in accuracy for the least investment in time and materials. Applying improvements in the recommended order provides the largest return on investment. Each step in the sequence builds on the solid foundation provided by the prior step. Maximum accuracy should achieved at each level before moving on to the next technology.

Completing all levels is not required. Once the accuracy goals are met, it is not necessary to proceed to the next level. The investment of time and materials significantly increases in relation to the potential improvement in accuracy from one level to the next.

4.1 Mechanical Accuracy

From a raw position error perspective, repeatability is the most critical error to understand. No electronic compensation can adequately correct for a machine repeatability problem. Repeatability starts with a mechanically sound machine.

4.2 Pitch Error Compensation

On a repeatable machine, pitch error compensation is the primary compensation feature that can significantly improve part accuracy, practically eliminating ballscrew pitch and backlash errors.

Again, machine repeatability is critical as no compensation can correct for mechanical repeatability problems.

Pitch error compensation corrects each axis individually and thousands of points can be used to fine tune positional errors. For the best per axis accuracy ensure that the machine is using bi-directional and interpolated pitch error algorithms. There are many resources available within manufacturing facilities and within the industry to properly set up pitch error compensation.

4.3 Setting Accurate Kinematics

For machines with four, five, or more contouring axes, an often-overlooked source of error is the kinematic dimensions. Kinematics is the relationships between the linear and rotary axes. Many machines have the machine tool builder's nominal mechanical print values in the CNC parameters for the kinematic. Kinematic errors exist due to geometric inaccuracies of structural components in a 5-axis machine tool and they can cause significant tool position and orientation errors with respect to workpiece that in turn lead to geometric inaccuracies of the machined surface in cutting operations.

Since the tool length is part of the kinematics on a 5-axis machine, it is equally important that the tool length values are accurate. Accurate kinematic and tooling dimensions are extremely important to the interpolation algorithms in the CNC. Frequently checking and calibrating these dimensions is required in order to maintain a high level of part accuracy. If these values are incorrect, no compensation system can correct for the error at the tool tip.

These values must be accurate both in the CNC and in the CAM system – and the effective values must be the same to generate and execute programs correctly. With vector programming, these values only need to be accurate in the CNC and they can be maintained during routine preventative maintenance.

4.4 Basic Volumetric Compensation

For the larger machines common in aerospace manufacturing, additional mechanical errors include straightness, squareness, beam sag for gantry style machines, and other errors.

A simple volumetric compensation system like FANUC's 3DEC can correct for these remaining mechanical errors (figure 13). Volumetric compensation values are more complex and costly to measure than pitch error compensation, especially when trying to gather small increments. It is more cost effective to first implement pitch error compensation for each axis and then it is easy to compensate for straightness, squareness and beam sag with a few volumetric points.

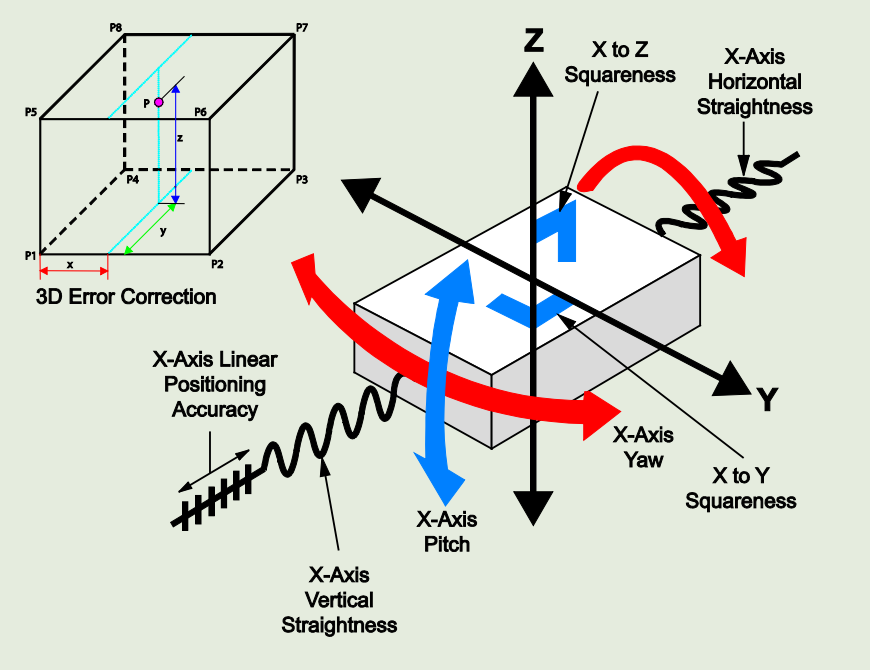


Figure 14. Degrees of freedom and common errors that can be compensated with 3D Error Correction

4.5 Thermal Error Compensation

After all the basics are covered, additional accuracy improvement requires a careful evaluation of thermal error. Thermal error analysis and correction is not a trivial task. The two main CNC compensation techniques are a) measurement of temperature at strategic locations with appropriate system offsets and b) direct measurement of physical displacement at strategic locations with appropriate system offsets.

Correcting for thermal errors in a 5-axis machine is very challenging, since the primary concern is the position of the tool on the workpiece. Algorithms that translate the thermal growth of a particular machine component to its effect on the movement of the tool tip are very complex and most likely impractical.

Improving 5-axis part quality using thermal compensation (or advanced volumetric compensation) requires a significant investment for a minimal return in accuracy improvement. Before considering implementing thermal compensation techniques, all efforts to normalize the machine and machining environment should be exhausted.

4.6 Advanced Volumetric Compensation

Larger 5-axis gantry style machines, may have additional roll, pitch, and yaw error throughout the work envelope. If all other efforts to reduce potential error in a system are resolved, an advanced volumetric compensation system similar to the Fanuc 3DREC compensation is a viable option.

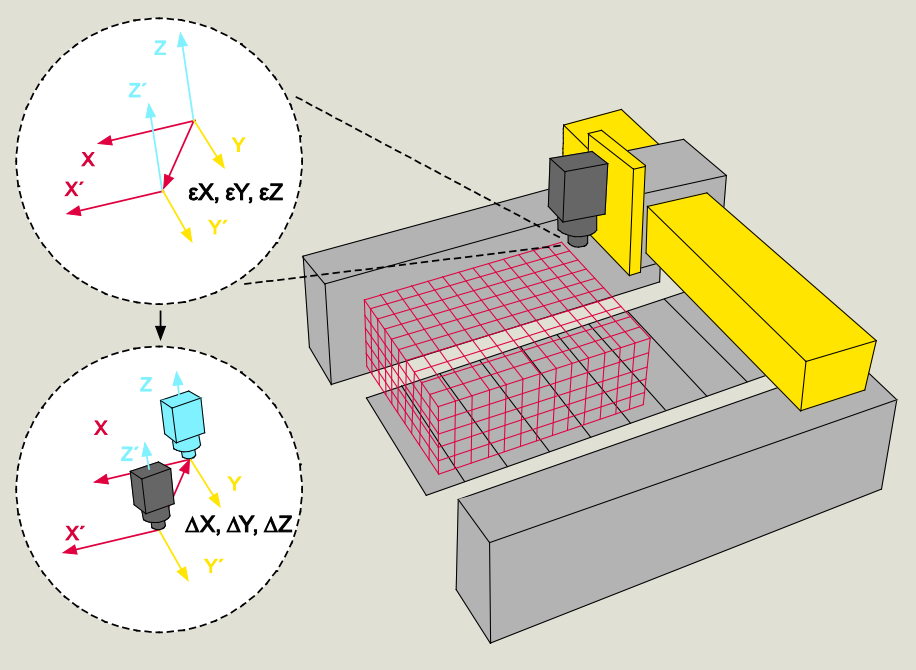


Figure 15. Volumetric compensation compensates for positioning errors in the complete machine work envelope.

For any advanced volumetric compensation system, additional highly-specialized measurement equipment is necessary to make the measurement process practical. In North America, examples of the required measurement equipment include the API Active Target system or the Etalon Trac-Cal system.

When trying to achieve the best possible accuracy from a machining process, all aspects of the process requires evaluation. Starting out with an advanced volumetric compensation system without addressing the more basics compensation issues will result in significant money spent for a minimal return.

With advanced volumetric compensation systems, it is important to keep in mind the required maintenance to maintain the newly achieved level of accuracy. Newly installed machines settle, machine use causes wear and mechanical changes over time, and machine damage such as a crash can dramatically change the resulting motion. Any of these events effectively invalidate existing compensation.

To achieve a very small improvement in accuracy using advanced volumetric compensation requires a disproportionate amount of effort and ongoing maintenance. An advanced volumetric compensation system requires a complex process and expensive measurement equipment that must be used to recalibrate the compensation system on regular intervals.

If the goal is to improve overall part accuracy, the entire manufacturing system must be evaluated. With a logical and thorough approach to improving the accuracy of the system as a whole, sustained improvement in part accuracy is possible.

5 High-Speed Roughing

High-speed machining is generally focused on finishing operations, cutting complex surfaces fast with high accuracy and a superior surface finish. Many large 5-axis parts are machined from solid blanks, and a significant amount of material must be removed before the finishing passes can be made. Many materials are difficult to machine, and tool wear is a common problem, limiting machining speed.

Part programs are typically generated by CAM or conversational programming systems using conservative feed rates, and then manually optimized by an operator adjusting the feedrate override on the CNC. Manual optimization can partially improve speed but it also prolongs setup time, reduces setup part yield and increases operator labor costs.

The maximum material removal rate for a machine is a function of the width of cut, the depth of cut, and the cutting feedrate. It is also a factor of the material type, and the condition of the tool. Unfortunately, many of these factors vary significantly during normal machining.

There are several sources of variation of width and depth of cut. The initial machining cuts on a rough sawn or cast billet may vary depending on the consistency of an upstream processes. A simple facing operation will typically have a final pass with a cutting width that is less than any of the previous cuts. And in contouring mode, the width and depth of cut can be changing continuously as the tool follows the shape.

As a tool wears, it consumes more power to remove the same amount of material. And if a tool is not loaded, due to any of the other sources of variation, the tool will start to rub rather than cut. Premature and uneven tool wear will be realized.

Since most of these variables cannot be known ahead of time and most will vary significantly during machining, the part programmer must assume worst case conditions, and specify a conservative feedrate, under-utilizing the machine.

Figure 16. Premature and uneven tool wear possible when tool is not loaded consistently.

One solution is real-time adaptive control. By monitoring the spindle power, adaptive control can detect when cutting conditions are varying. If material hardness, width of cut, depth of cut or the tool wear factor increases, spindle power will rise, and the axes feedrate is reduced to maintain the power level. If any of the variable factors are reduced, spindle power will drop, and feedrate is increased to sustain the set power level.

In tests, adaptive control has reduced cycle times by more than forty percent. It typically extends tool life by keeping tools loaded and eliminating "rubbing". It also reduces labor costs by eliminating manual optimization and minor stoppages related to premature tool maintenance.

Though the concept of adaptive control has been around for many years, advances in high-performance CNC hardware makes its wide-spread use more practical. Adaptive control is a valuable roughing complement to the various high-speed finishing technologies for many 5-axis applications.

Figure 17. Adaptive control during roughing complements high-speed machining during finishing.

6 Summary

It is clear that that the traditional machine-centric 5-axis workflow is not competitive and that any new equipment purchase or machine retrofit is an ideal time to make the transition to the contemporary part-centric workflow. Existing equipment may already part-centric workflow capable and it is just a matter of using the CNC features appropriately.

Case studies show that the part-centric workflow delivers significant improvements in cycle time, part accuracy, and surface finish. Using tool center point control, modular programming and vector programming delivers the ultimate in part program portability between widely different machine tools, increasing schedule flexibility and reducing wasted machine capacity. The post processor is significantly simplified, reducing cost and making it easier to add additional machines and applications.

Quality improvements begin with a part program that is a true definition of the part, uncontaminated by machine specific inaccuracies. Using the point-to-point approximation as a model of a complex curve rather than the desired tool path allows smoothing and fairing to recreate the curve. Using this method, the machined part is more faithful to the original CAD model and it can be machined at a higher feedrate, producing a better surface finish and with less stress on the machine mechanics. Tool center point control is essential to prevent minute workpiece gouging that shows up as an inferior surface finish. A suite of compensation tools are available to improve the accuracy of a mechanically sound, repeatable machine, but they must be implemented in the correct order to deliver the optimum results.

Real-time adaptive control and the material-condition-select function combine to optimize feedrates, axis performance and accuracy requirements to maximize the material removal rate during roughing and then re-optimize for accuracy and surface finish during finishing operations.

Quick setup, gouge-free machining, fast and smooth complex curves, programmable axis response, flexible modular programming, improved part accuracy and high-speed roughing are just the highlights of what is possible with the modern part-centric 5-axis machining.

FANUC America Corporation
1800 Lakewood Boulevard
Hoffman Estates, IL 60192
888-FANUC-US (888-326-8287)
Find more information at www.fanucamerica.com

Technical data is subject to change without prior notice. No part of this document may be reproduced in any form. All rights reserved.
© 2015 FANUC America Corporation

Document # MWA-030-EN_01_1511