HARTWIG

Your Guide to CNC Machine Tool Controls:

FANUC vs. Okuma OSP

Nearly every industry uses CNC milling machines, but not all CNC milling machines are created equal. A large difference are the controls that run it. In this whitepaper, we'll explore the differences between two controls for CNC machines: FANUC and Okuma's OSP. We'll discuss the strengths and weaknesses of each system, and how they interact with the machines, as well as the different aspects that make them distinct from one another to help inform your decision when selecting your next CNC machine.

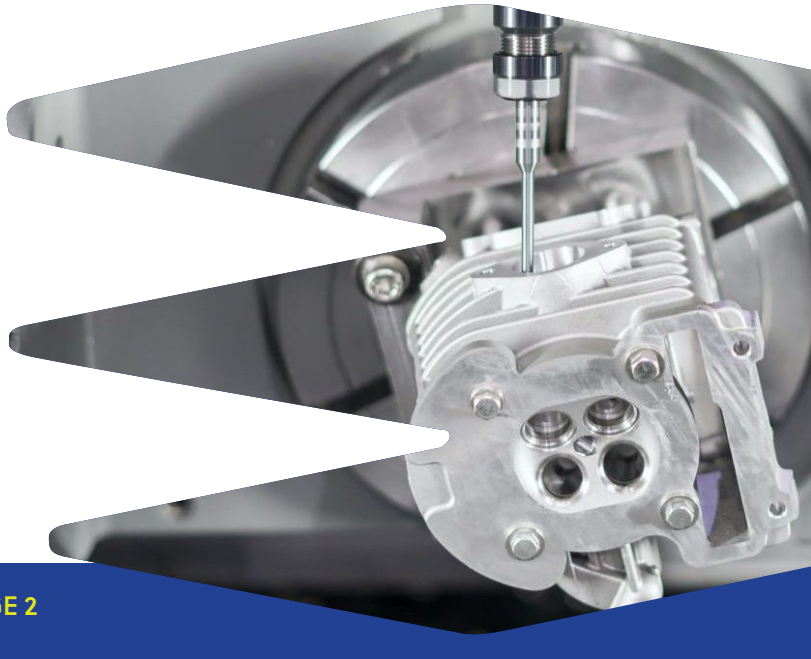
Visit just about any machining facility, from major assembly lines to small tool-and-die shops, and everywhere in-between, and you'll find CNC machine tools in use. Just about every industry uses them, including typical "job shops", high-end scientific labs, aerospace, and automotive industries. CNC machining centers and lathes produce complex parts accurately and with a high degree of consistency.

Not all CNC machines are created equally, and one significant differentiator is the control that runs them. In this article, we compare the important differences between two control systems in the market, FANUC and Okuma's OSP, to provide insight to help better inform buyers for the future purchase of CNC machines.

Okuma's CNC control technology spans more than 50 years. When Okuma made the decision to build their own controls in the middle 1960s, they became unencumbered by what other machine and control builders were selling. Okuma took advantage of having a clean sheet of paper upon which to start.

The current Okuma OSP control is built using a Windows®-based, open architecture platform, making it easy to install PC applications that allow a number of processes, including access to tooling lists/setup spreadsheets, and process documents, among others. The open architecture platform also allows users to continually customize operations to suit individual CNC machining requirements. Think of it this way:

Most of us have the APPs we need for our personal and professional lives installed on our smartphones.



What are the differences for the CNC operator?

Because OSP uses a process-based user interface rather than the traditional mode-based operational layout, the platform centralizes data in one easy-to-access screen, rather than moving back-and-forth from one location to another.

Positioning

A CNC (Computer Numerical Control) control commands and monitors machine movements. The overwhelming majority of these machines use ballscrews to move the tables and slides on a machine. Generally a command in the program is converted to movement by the control, using a "drive" to send signals to a "servo" (motor) which is connected to the ballscrew. This is consistent for all controls, but it is here where the difference between and OSP and a Fanuc begins.

A Fanuc use pulses, generated by a pulse generating device, that are sent back to the control from the servo/ballscrew. The control counts the pulses to know where the machine's slides and tables are, and monitors the frequency of the pulses to check the speed at which the slides are moving. The control compares where the slides are to where they should be and makes the necessary adjustments.

At any given moment the axis position are stored in the control. That portion of the control is provided power by batteries with the intent that when the control is turned off, the control will know where the axis are when the control is turned on, eliminating a need to re-establish each axes position. This system works well, **EXCEPT** if the control is turned off, either intentionally or by mistake, when the batteries are bad.

The OSP controls use Okuma's patented Absolute Position Encoders. These electro-magnetic encoders send the actual position data to the control. The control then has less math to do, but the very significant advantage is the axes always know where they are without the control having to keep track. Power failures, bad batteries, etc. have no impact on the machine/control's ability to know where the axes are. Even if the control is off, when it is turned back on, it will immediately know where the everything is located. The benefit? Crashes due to a lack of referencing or "zeroing" are completely eliminated.

Programming and program operations.

The standard language for CNC control is the "Word Address" format, which consists of "G" codes that represent axis position commands (X, Y, Z, etc), and "M" codes to turn functions on and off. Generally, the primary codes for cutting, drilling, RPMs, and feed rate are the same. More control specific codes and advanced functions are different. Only as the complexity increases, do the differences become more obvious.

One example is in macro (sub-programs) programming. Fanuc uses M98 to jump to a sub-program, Okuma uses the word "CALL" (a bit more computer like).

Fanuc has used numbers for variables. #2201-#2400 is Z-axis tool geometry. Okuma uses VTOFH[1] for Z axis geometry (length) as a primary tool in machining centers The convenience is the Okuma variable can be read: Variable Tool OFfset, H making it easier to remember or understand when looking at a program.

Okuma stores programs either onto bubble memory (old controls) or onto a hard drive (newer control). Programs can be named with letters and/or numbers so they a represent a part number or a customer's name, not limited to a letter "P" followed by a number. When a program is selected, a copy is put into RAM memory (except for very large programs). The machine runs the copy in RAM memory, leaving the orginal for editing. No optional or separate "Background Editing" feature is needed. The program can be left in edit mode for as long as needed. Once the program is running, no editing is required. This can save a significant amount of time and headache.

Since the early 1980s, all OSP controls have cut and paste capabilities. The newer controls provide the ability to run two programs simultaneously for editing while the machine is running, allowing users to cut and paste between the two programs.

Conversational Programming

Not only does Okuma's OSP control run on the Word Address format ("G-code"), but Okuma has a powerful and effective conversational control for both machining centers and lathes. This is named AOT for Advanced One Touch, and is standard on many lathes and machining centers. Machines equipped with AOT can also use the standard coding, so there are no limitations to the programming methods used. In the past the OSP's conversational portion output a standard G-code for the machine to use. What a great learning tool! Some newer models will also run on the conversation data without generating a G-code program. These models look at the speed and feed overrides. If a G-code program is created, the RPMs and feed rate will be output accordingly.

High Speed Machining

Both Fanuc and Okuma's OSP machining center and multi-function lathe controls have penetrating block look ahead for high speed, multi-axis machining. The format can use either the standard G-code or NURBS G-code or portions of the program can use NURBS. And like others, the OSP control can use tolerance to control the feedrates both from program input or parameter input, but, if given the same parameters used in the CAM system, can also re-construct a surface back to its original designed shape.

In conclusion, understanding the differences between the two operating systems, and the machines they work with is integral to finding the right solution for your next project. Establishing which system works with your staff's expertise, programming skills and project complexity will ultimately help you make the right choice to drive strong results from day one.