



Machining

The Advantages of 3+2 Machining on 5-Axis Machines

Kip Hanson | May 08, 2018

What You Need to Know

Using a 5-axis machine eliminates two-thirds of the setups and part handling, so fewer operations mean fewer fixtures, reduced WIP and less scrap.

A 3+2 "positioning mode" on a 5-axis machining center is not limited to only 90-degree movements. Five-axis machining centers offer control features not found on their 3-axis cousins. Volumetric accuracy is assured with capabilities that automatically compensate for and adjust kinematic inaccuracies. There are a host of tool holders, workholders, clamping, vises and zero-point systems needed—some that are specifically designed to make it very easy to remove and replace parts in the machine.

From easier, faster setups to tighter tolerances and better control options, performing 3+2 machining on a 5-axis system can help ensure some of the most accurate and efficient part-making in today's manufacturing.

There are many reasons for using 3+2 machining on a 5-axis machine. Maybe the big contract for the medical implants your shop used to justify the purchase of a 5-axis machining center ended and you're looking for new work. Perhaps you're having a hard time meeting true position requirements between two part features that are currently being machined on separate operations. Or maybe you're tired of all the parts waiting to be completed on your 3-axis machining centers. Whatever the situation, using your 5-axis machine to do 3+2 work could be the best decision you've ever made. Here's why.

Adding Up the Advantages of 3+2 Machining

Need convincing? Let's review the setup, tolerance, metal removal and flexibility that can be accomplished with 3+2 machining on a 5-axis machine.

Assuming your parts are orthogonal and need machining on all six sides, which is common for most milled parts, a **5-axis machine** eliminates two-thirds of the setups and part handling. Fewer operations mean fewer fixtures, reduced WIP and less chance that a part will end up in the scrap bin along the way. It also means less overproduction in anticipation of lost parts during the manufacturing process. Instead of five or six extra parts, just one will now suffice.

Geometric tolerances between features on different sides of the workpiece are easier to hold when completed in the same operation. The ability to tip parts at whatever angle you desire often means tools can be made shorter and therefore more rigid, improving tool life and part quality while increasing **metal removal** capabilities. Because it's easy to rotate around to multiple sides of the workpiece in a single operation, there's greater flexibility in the sequencing of the machining steps. Deburring of intersecting features is much simpler as well.

A 3+2 "positioning mode" on a 5-axis machining center is not limited to only 90-degree movements. The angled holes in the hydraulic manifolds you've been working on can now be completed without an additional setup or fixture. The same goes for the threaded boss on the F-16 valve body that sits at an awkward 27 degrees, or the tapered and undercut sides of the mold you just quoted. These features are easily produced on a 5-axis machining center, even if you never do use it for "true" 5-axis work.



FORUM

What's your take? Talk to your peers *in the community forum*.

The Software Side of 3+2: CAM

Perhaps the biggest challenge you'll face when adopting 3+2 machining practices is the programming. Collisions are more likely when part zero planes are constantly changing, so care must be taken to clear the workpiece during indexing and when reaching deep into pockets and slots.

Here again, though, there's little to worry about. Even a basic CAM package supports 3+2 machining—simply assign a work coordinate system to each side of the part and program as you normally would. But most CAM systems offer more advanced 3+2 support than that, even without a simultaneous 5-axis milling module. Multi-faced parts are easily programmed by selecting whatever side you want to work on and letting the software take care of the indexing commands. Visualization of the machining process is improved, collisions are more easily avoided, and air-cutting is minimized.

Benefits aside, some argue that 3+2 cycle times are longer overall compared to the traditional technique of machining a fixture full of parts on a VMC, one side at a time. Maybe not. A good CAM system takes full advantage of each cutting tool, machining every side possible in one operation and drastically reducing the number of tool changes and wasted motion. Granted, there's more to think about and the program will be longer, but any mistakes you might have made on Op. 5 will now be made on Op. 1, potentially saving you big bucks.

Need more information on CAM systems? Read "How to Plan for and Invest in a New CAD/CAM System."

3+2 Machining: Primed for Accuracy

Despite the clear advantages, however, some shops remain reluctant to pursue a 3+2 machining strategy on a 5-axis simultaneous machine tool. It might be considered overkill, for example, to use one of the most capable and technologically advanced machines in the shop in this manner, especially compared to installing a tilt-rotary trunnion table on a 3-axis vertical machining center.

Yet, 5-axis machining centers offer control features not found on their 3-axis cousins, trunnion or not. Volumetric accuracy is assured with "auto tuning" capabilities, which automatically compensate for and adjust kinematic inaccuracies. Tool center point control and tool vector input simplify programming, and dynamic work offsets simplify setups. With their advanced controls, five-axis machines are, quite simply, better at multisided machining, whether used in simultaneous mode or not.

Cool, but now what? What about the workholding and the tool holders? Must a shop make even more investment in tooling once the decision's been made to go 3+2? The good news is that—provided the spindle taper is the same—all of the tool holders currently on your 3-axis VMC can be used on a 5-axis machine. So, too, can be the vises, fixtures and chucks, although a few caveats exist.

It's Tool Time: What You Need for 3+2 Machining

Let's start with the *tool holders*. Anyone buying a new machine tool, whatever it is, should leverage the opportunity to maximize its potential with high-quality tooling. If an HSK or similar dual-contact spindle is available, then by all means get it. The greater speed and accuracy of a modern interface more than makes up for the additional tooling cost.

Go deeper. Check out "Lessons in High-Performance Machining: Don't Forget the Tool Holders."

Also, this might be the perfect time to ditch the sidelock and collet chucks you've been using for *shrink-fit* or *hydraulic holders*, thus keeping pace with your machine's capabilities.

On the workholding side, life just became much easier. A wide assortment of 5-axis *vises* are available, most of which grip the bottom 1/8-inch or so of the workpiece and lift the part high enough to provide full access to all sides. When the part's done, just slice away the sacrificial material and machine whatever remains on the bottom of the part.

While you're at it, why not equip your new 5-axis vises with a zero point or comparable quick-change clamping and positioning system? Unlike a 3-axis VMC, where the entire table can be filled with parts, machining with 3+2 generally means one workpiece at a time. A zero-point system makes it easy to get parts in and out of the machine quickly, and it reduces setup times to boot.

How do you use 3+2 machining to your shop's advantage? Do you use it on a 5-axis machine? Share your tricks and tips.

www.mscdirect.com/betterMRO

Copyright ©2024 MSC Industrial Supply Co.