

High Speed Machining Info

We're all learning this together so please let us know if you've got more info on this that you can share. Thanks to everyone who supplied info and it should be noted that much of this info comes from the Fanuc and Mazak manuals.

Advanced Preview (LOOK-AHEAD) Control

G08P1 - 2-2 1/2 Axis work is where Look Ahead is best suited. It can help with 3D programming but it just does not have the speed to keep up with large amounts of data and small moves.

This function is designed for high-speed precise machining. With this function, the delay due to acceleration/deceleration and the delay in the servo system which increase as the feedrate becomes higher can be suppressed. The tool can then follow specified values accurately and errors in the machining profile can be reduced.

AICC and AI nanoCC (Artificial Intelligence Contour Control)

2 1/2 - 3 Axis Work is where AI-NANO 1 Shines.

G05.1Q1 - Simple HPCC - 15 block look ahead

G05.1Q2 - Smooth Interpolation - filters and creates arcs in the control, 40 block look-head (200 block look ahead for nano)

It has more look ahead capabilities, it processes faster than Look Ahead and can better cope with large amounts of data with small moves. The Q2 function has the ability to filter and create arcs for a changed (smoother) path shape and therefore faster machining

To call AICC the following must be modal

G40

G49

G69

G94

G97

You must use G94 for the Fanuc functions. You cannot change to G95.

AI-HPCC and AI-nanoHPCC (High Precision Contour Control)

G5P10000 - HPCC (On older controls) AI-NANO 2 (On newer 31i Controls) is most definitely suited for detailed moldwork. Huge amounts of data with very small moves. This is a hardware option that involves RISC Processors. If you do Moldwork, this is the option for you. You won't really see any benefits doing 2-2 1/2 axis work with this option. 500 block look-ahead.

Some machining errors are due to the CNC. Such errors include machining errors caused by acceleration/deceleration after interpolation. To eliminate these errors, the following functions are performed at high speed by an RISC processor. These functions are called high-precision contour control functions.

(1) Function for multiple-block look-ahead acceleration/deceleration before interpolation. This function eliminates machining errors due to acceleration/deceleration.

(2) Automatic speed control function which enables smooth acceleration/deceleration by considering changes in the figure and speed and allowable acceleration for the machine. This is performed by reading multiple blocks in advance.

G43 cannot be specified in HPCC

Pretty much all G and M codes, even T codes, sub calls, etc. are allowed inside this mode

To cancel cutter comp G40 you must have motion before cancelling HPCC

To call HPCC the following must be modal

G40

G69

G94

G97

Mazak Geometry Compensation/Accuracy G61.1

The geometry compensation function (G61.1) is provided to reduce conventional geometry errors caused by delayed follow-up of smoothing circuits and servo systems. The geometry compensation function is canceled, or replaced, by the functions of exact stop mode (G61), automatic corner override (G62) and cutting mode (G64).

In the mode of geometry compensation (G61.1) the feed of the tool is automatically decelerated at relevant corners and for circular motions by the optimal corner deceleration and the circular feed limitation, respectively, in order to enhance the machining accuracy. Specifying an accuracy coefficient in the machining program can further improve the accuracy by additionally decelerating the feed for the sections concerned.

Specifying an accuracy coefficient 1 to 99 at address “,K” increases the machining time according to the additional deceleration at relevant corners and for circular motions.

EG. For K30 that means the rate of feed for corner deceleration or circular motion will be reduced by 70%

You can use both of these Mazak functions in both G94 or G95 but you cannot change during.

Mazak High Speed Machining Mode - G05 P2

The high-speed machining mode feature allows high-speed execution of programs used for the machining of free-curved surfaces that have been approximated using very small lines.

In high-speed machining mode, microsegment machining capabilities improve by several times, compared with conventional capabilities. This allows the same machining program to be executed at several times the original feed rate, and thus the machining time to be reduced significantly. Conversely, a machining program that has been approximated using lines of several fractions of the original segment length, can also be executed at the same feed rate, so more accurate machining is possible.

Combined use of the high-speed machining mode and the shape correction function allows more accurate machining to be implemented.

There are a series of associated parameters for tweaking cutting feeds, acceleration, deceleration, and fairing (removing of protruding data points). Check these out in your Mazak manual.