Application Note: 5-sided & 5-axis machining

Introduction

Efficiency equals profit

In this competitive climate, identifying ways to be more efficient is the best way to increase your bottom line. This application note will discuss the productivity benefits of 5-sided machining and 5-axis machining.

Details

Think first. Do second.

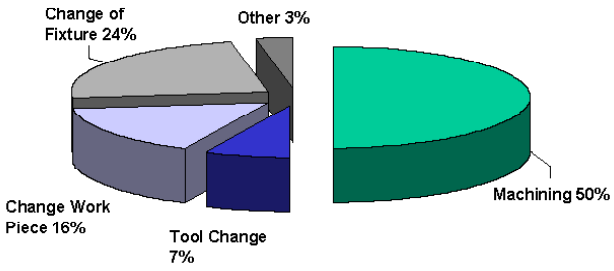
There are several ways to approach machining a part. This application note will compare examples to help you determine best practices. This application note will also explain control software features that increase productivity when machining complex parts.

PROBLEM: Smaller batch sizes. Reduced turnaround times.

SOLUTION 1: Evaluate the way you approach the part.
Example: One customer reduced their lead time by two weeks by machining the part from a billet instead of a casting. Machining a part from a billet eliminates the need to stock finished parts, eliminates delays due to lead times for castings, and eliminates the cost of storing castings and the labor costs associated with handling them.

SOLUTION 2: Spend money to make money. Upgrade your equipment.
Example: Investing in a machine with faster feed rates and more torque can significantly decrease cycle time. Old machine: 3 hours. New machine: 1 hour 30 minutes.

SOLUTION 3: Machine design affects efficiency.
The decreased cycle time our customer experienced in Solution 2 was due in part to the horizontal orientation of the part because it allowed the chips to fall away from the part which made it possible to run a 1-inch endmill at 450 ipm to rough out the pocket.



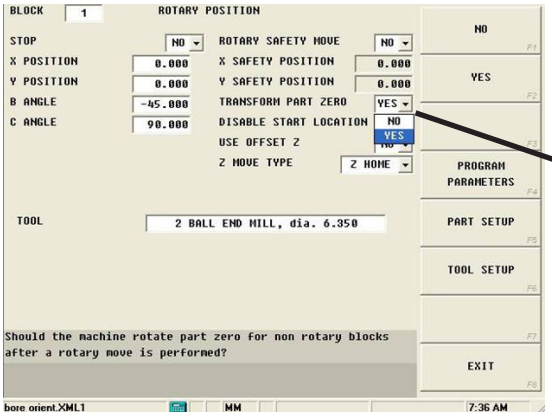
Capacity Chart
Data from American Mold Builders Association

SOLUTION 4: Reduce the number of times you have to flip the part. The Capacity Chart shows that 24% of a job's capacity is spent on fixturing. Although using an index table is preferred over using a row of vises that requires you to flip each part into the next vise, there are limitations to an index table solution (e.g., each time you handle the part to refixture it, you spend time blowing off the chips, unclamping the part, wiping off the vise, and clamping the part into the vise). Therefore, a palette changing system is often preferred because it keeps the spindle moving.

Evaluate the job: Number of parts and number of tool changes. Even though the Capacity Chart shows tool change at only 7%, tool change optimization may provide sufficient time savings if you have multiple tool changes. Consider at what number of parts does it make sense to use time to set up another fixture on a palette and locate that part versus flipping a part in a vise.

Application Note: 5-sided & 5-axis machining

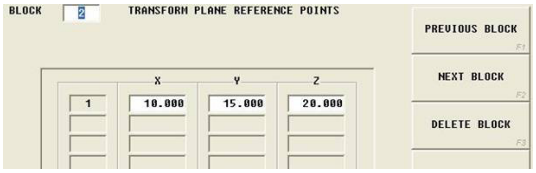
PROBLEM: Time wasted on multiple part zero setups. With complex parts, you could easily have five part zero locations.



SOLUTION: WinMax Transform Part Zero feature

Locate just one part zero and the control calculates the other locations for each of the rotary moves.

1. Create a Rotary Position data block
2. Set the Transform Part Zero field to YES



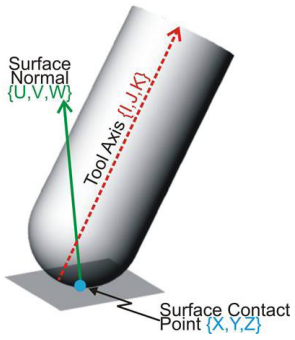
(EXAMPLE PART: X10, Y15, Z20)

3. Insert a Transform Plane Reference Point data block
4. Specify the distance to shift the part zero relative to the original setup

NOTE: Pattern Data Blocks can also be applied to transformed features.

PROBLEM: For 3D surfaces, you cut off of the tool centerline to get a better surface finish but you have to repost the program if you adjust the tool diameter for wear.

Workpiece surface contact point + contact point surface normal + tool vector



Example

M128 3D Tool Geometry Compensation
G41.2D_ R_
G00 X0.Y0.Z0. U0.V0.W1. I0.J0.K1.
G01 X10.Y10.Z10. U0.V0.W1. I0.5J0.5K0.707106 F1000

XYZ = Surface contact point
UVW = Surface normal at contact point
IJK = Tool vector

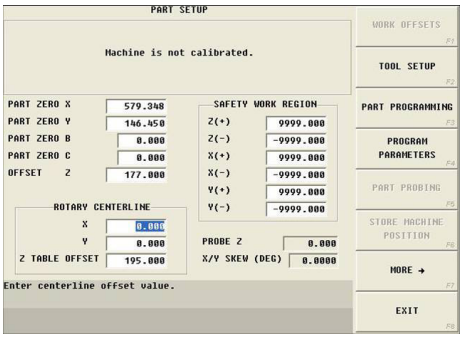
SOLUTION: WinMax 3D Tool Geometry Compensation

This feature makes it possible for the control software to compensate for the tool geometry. Position the tool off of its centerline while cutting and adjust the 3D radius offset of the tool. 3D Tool Geometry Compensation makes it possible to adjust a tool diameter and continue cutting the part without needing to repost the part program. In general, you wouldn't want to adjust the diameter because you would risk gouging the surface.

Application Note: 5-sided & 5-axis machining

PROBLEM: Complex and difficult post processors. CAM systems generally use tool vectors to generate the tool path. To make programs independent of the machine's rotary configuration and to simplify the post processor, a newly developed control software feature called Tool Vector Input has been developed for 5-axis machining centers.

SOLUTION: The Tool Vector Input feature lets you specify the tool tip location relative to the workpiece and the tool axis vector instead of using address letters to specify the B and C axes angles. Executing the program is much faster because the post doesn't need the machine configuration and the centerline of the rotary axes. WinMax 5-axis control software automatically compensates for the position of the tool in part coordinates relative to the center lines of rotation for the B and C axes. Therefore, the control understands where the work piece is located relative to the center lines of rotation of the rotary axis.



1 Fixture the part onto the machine

2 Locate part zero in X and Y

3 Touch off your tools to the calibration point

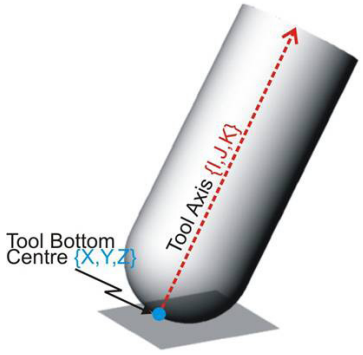
4 Determine the Z distance from the tool touch off point to the part zero

5 Determine the Z distance from the table to the part zero

(The control's software automatically compensates for the position of the tool in part coordinates relative to the center lines of rotation for the B and C axes.)

workpiece coordinates + tool vector
G01 X10.Y10.Z10. I0.5J0.5K0.707106
would be equivalent to:
G01 X10.Y10.Z10.B45.C45

- Tool Vector input makes programs machine independent
- Control computes machine angles and positions
- Simplifies post-processor



PROBLEM: Redundancy in posting a five-axis program. Each time you refixture, you have spend valuable time re-entering the distance from part zero to the centerlines of rotation and repost the part program.

SOLUTION: Tool Center Point Management

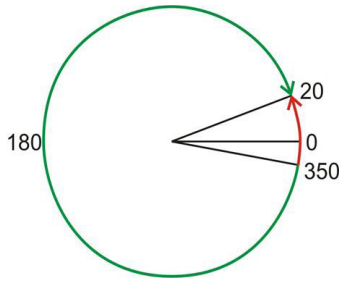
Not only do you save setup time with the Tool Center Point Management feature of Hurco WinMax Control software, you save time you would waste reposting the program. Post the program once and machine the part, no matter where it is in relation to the centerlines of rotation on the machine. M128 solves the problem for CAM software. The CAM programmer generates the tool path based on the part model's zero location. Therefore, you can post the program independent of where the stock is fixtured on the table–a substantial time saver for a five-axis part.

Application Note: 5-sided & 5-axis machining

PROBLEM: Rotary axis takes the longest angular path to the target position.

SOLUTION: M126 Shortest Angular Traverse Function

The rotary axis on any machine is generally the slowest. With M126 you are able to command the machine to take the shortest path to reach the target position.



Example: If you position the rotary axis at 350 degrees and then command the next position to 20 degrees with the M126 Shortest Anular Traverse turned on, the distance traveled will be 30 degrees.

PROBLEM: Marks on the part. Software-generated tool paths can result in odd looping rotary moves that leave marks on the part when the program is interpolated by the machine's control.

SOLUTION: Tool Path Linearization

At Hurco, we have devoted a great deal of engineering resources to find a solution. This led us to develop a feature called Tool Path Linearization, which is specific to 5-axis, G-code programs. Tool Path Linearization eliminates the many line segments in the form of XYZBC or AC moves that a CAM system uses to create a smooth part. With tool path linearization, the tool tip and tool vector can be interpolated between tool positions with respect to the workpiece even with the tool and part rotating inside the machine.