surfcam

: waveform

A Primer for Setting Proper Feeds and Speeds

The new Waveform cycle is superior to traditional roughing cycles, where machinable geometry features are offset (inward or outward) by the % of stepover. Traditional toolpaths must often run slower feeds and speeds due to the variable width of cut conditions encountered in corners. Tool load spikes as chip thickness increases in areas where the tool finds more material. Waveform toolpath solves this problem.

Waveform Roughing :

Reduces cycle time

Improves tool life

Lengthens machine maintenance cycles

Keeps constant chip load

Cuts deeper and faster

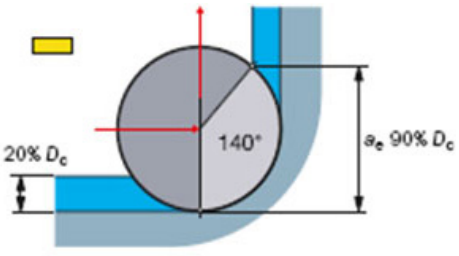
The Waveform roughing toolpath has been developed to remove tool load spikes and maintain consistent chip load. Traditional roughing toolpaths result in inconsistent tool load and erratic machine motion. Waveform's algorithm calculates toolpath based off the actual width of cut, generating a fluid toolpath throughout the machinable elements using a smooth flowing motion, without the need to program feeds and speeds for heavy tool load conditions.

Consistent tool loads generated from the Waveform toolpath offer users the opportunity to completely rethink speeds, feeds and depth of cut. Continuing to use your standard speeds and feeds will work, but are likely to be very conservative. Traditional toolpath speeds and feeds were calculated with consideration of extreme cutting conditions. Consistent tool loads allow much higher feeds, speeds and full depth of cut, permitting material removal at the tool manufacturer's recommendations.

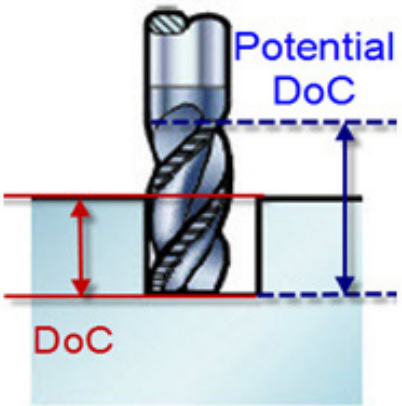
Cutting Data Calculations

Straight line test cuts help maximize your cutting potential because they mimic the Waveform toolpath. Calculating the correct speeds and feeds takes minimal testing to generate. The information below will help you to set up your own straight line tests and discover your optimized roughing potential.

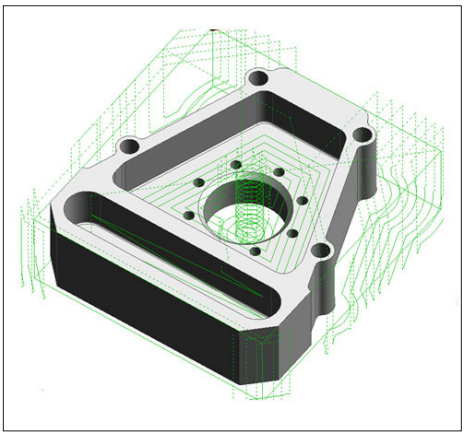
Cutters with specific flute lengths can be utilized to obtain even wear throughout the length of the tool. Spindle speeds and feeds are optimized using basic testing methods that can vary depending on work holding, tooling, fixturing, machine and material. Speeds and feeds are typically faster than traditional toolpath, while the depth of cut typically starts at 1 to 1 1/2 times the diameter of the cutter.



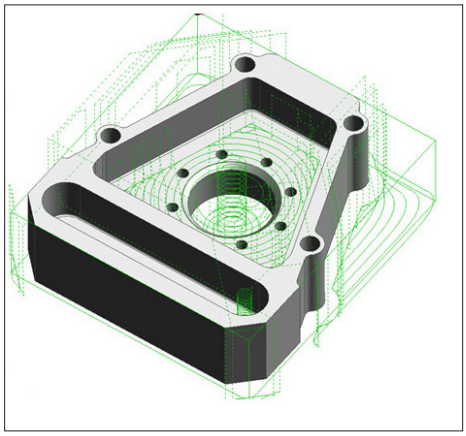
Traditional - Toolpath Overloading in a Corner



Waveform machining is standard with Surfcam. No additional purchase necessary.



Traditional Toolpath



Waveform Toolpath

Starting with tooling, using an odd number of flutes helps with rigidity. Recommendations are to use 5 to 7 flute cutters on Steel and 3 to 5 flute cutters on Aluminium. Using hydraulic, shrink fit or collet chucks are highly recommended. Weldon chucks are not recommended because they don't encompass the full diameter of the cutter, throwing the cutter out of balance.

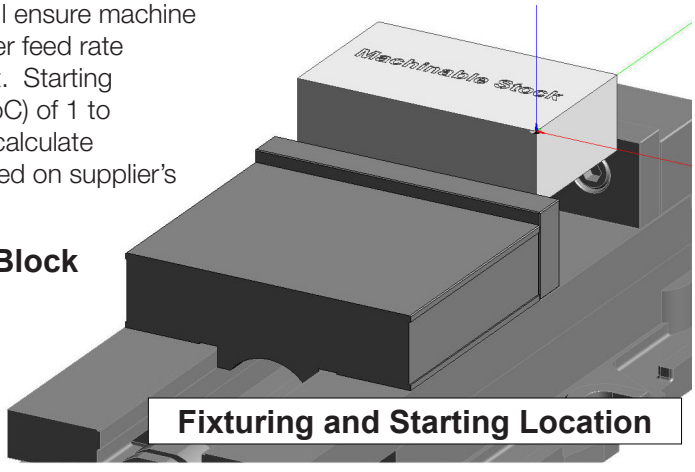
Place the test material in your machine's fixture, leaving a machinable amount of stock above it. Assigning the work offset at the lower right corner will simplify the NC code. Looking toward the machine, we will test cut the front of the stock from right to left, creating a straight line climb cut. A large lead-in will ensure machine acceleration to a proper feed rate before entering the cut. Starting with a depth of cut (DoC) of 1 to 1 1/2 times diameter, calculate speeds and feeds based on supplier's recommendations.

Test Block

Sample Straight Line Test Code

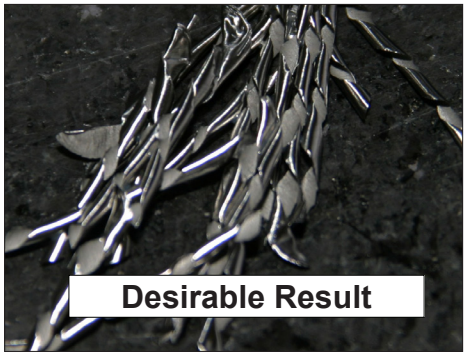
```
%O1000
N1 G90 G20 G00 G40
N2 T1 M6
N3 S[RPM] M3
N4 M8
N5 X3. Y[%Stepover X Tool Dia]
N6 G43 Z0.25 H01
N7 Z[Depth of Cut]
N8 G1 X-10 F[Feedrate]
N9 G0 Z0.25
N10 G28 Z0
N11 G28 X0
N12 M30
```

Key factors to consider for straight-line test cuts are chip colour, chip edges, load meter, and sound.



Fixturing and Starting Location

Proper chips should have a smooth edge from start to end (see photos). In Steel, heat from machining will be removed with the chips, leaving them a bluish brown color.



Desirable Result

Undesirable Result

Based on your tool type and material, adjusting one parameter at a time for depth of cut or width of cut will optimize feeds and speeds.