



Surface Finish Tips

Scan code to get the latest version of this document



Translation Available



INTRODUCTION

- This document tells you about the different tools that you can use to machine a better quality surface finish. We used all of the topics in this document to produce the part shown below. We used the correct tool path, the correct cut parameters and tolerances in the CAM system, and had the correct parameters and features in the machine control.

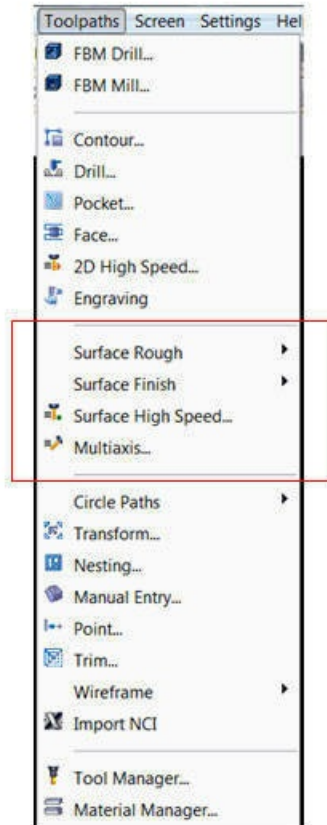
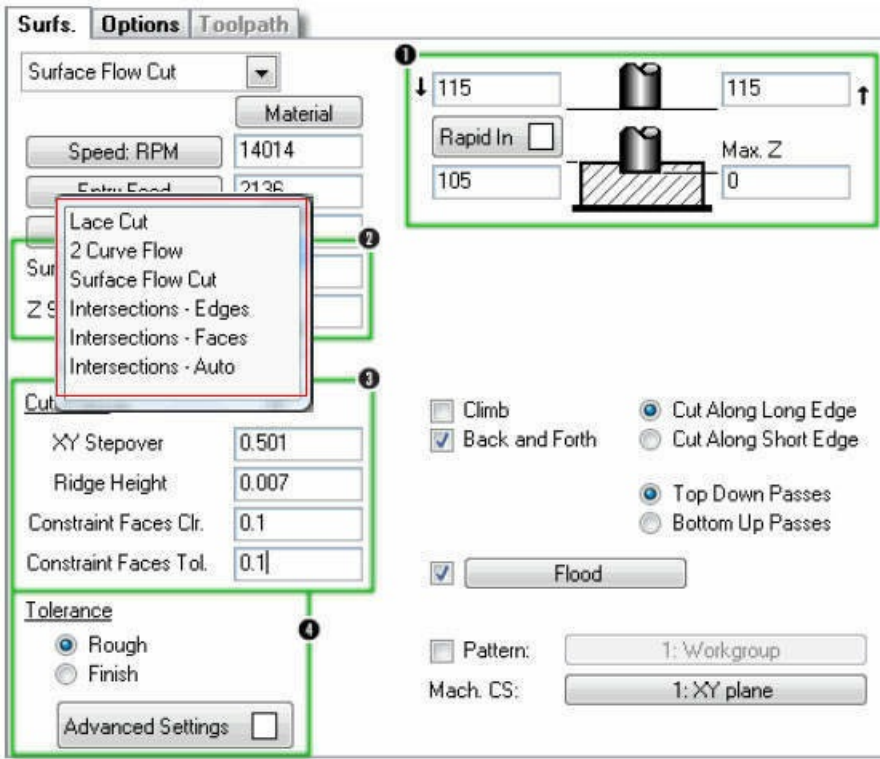
Tool Path Types



For roughing, typically the newer Dynamic (constant engagement) tool paths work the best.

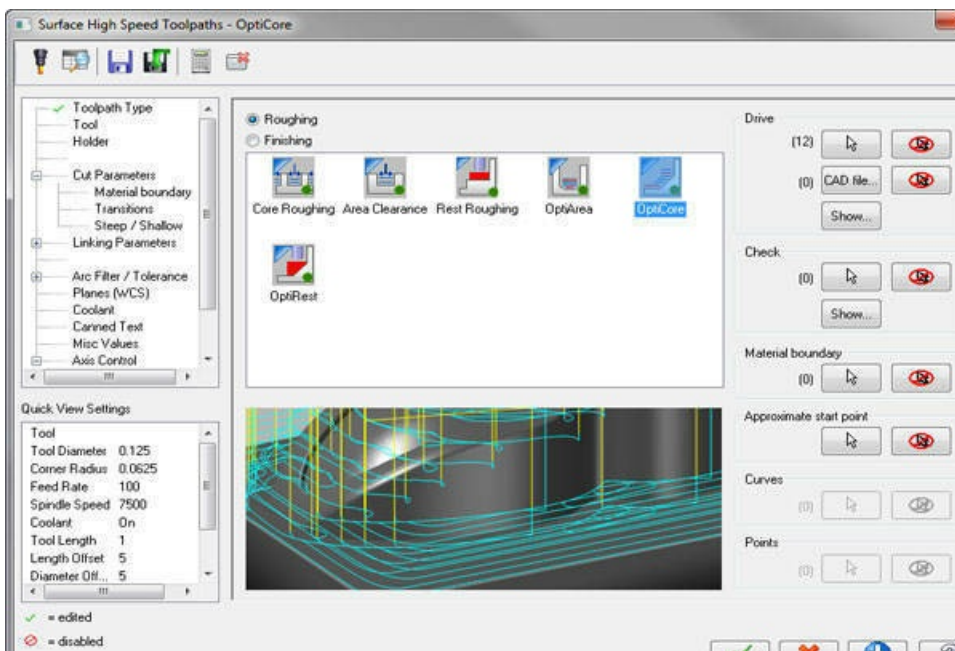
For finishing, select the type that will give you the best finish based on your part shape.

The tool path type may be different for different CAM systems.



The screen shot on the left is from Mastercam.

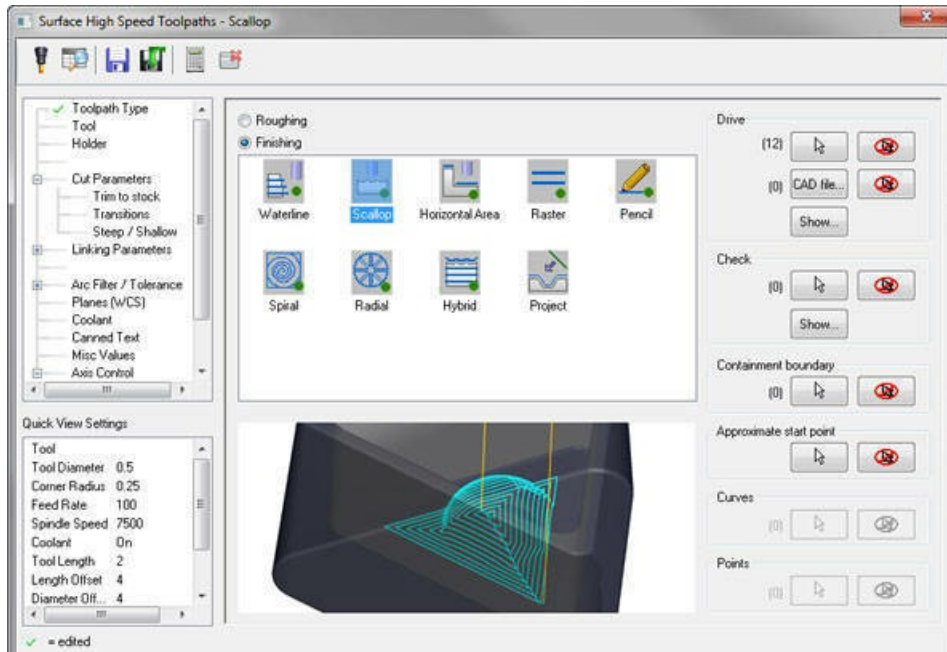
The screen shot on the right is from GibbsCAM.



Dynamic roughing tool paths like Mastercam's OptiCore and OptiArea are great tool paths to use.

They calculate the toolpath to have constant engagement in the cut, which lets you use higher feed rates and gives better tool life.

This lets you remove material at a higher rate, and you can save tooling costs.



When you select your finish toolpath, do not select a toolpath that changes direction on the part.

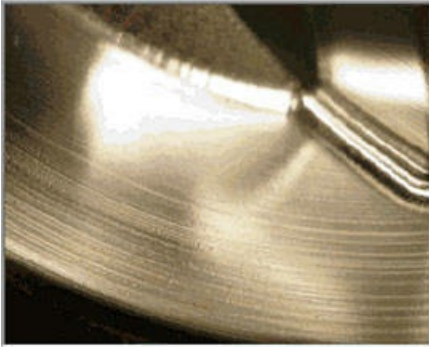
The scallop tool path changes directions on the part.

This change in direction leaves witness marks or seams on the part. This produces a visually undesirable finish.

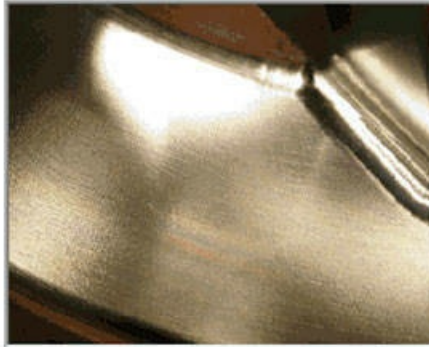
This is acceptable for semi-finishing. It is never acceptable when you need the best possible finish.

Select a tool path that changes direction off of the part. The shape of your part determines the path you need to use, and you might need many finish operations on different features of the part.

3D Tool Path Refinement



1



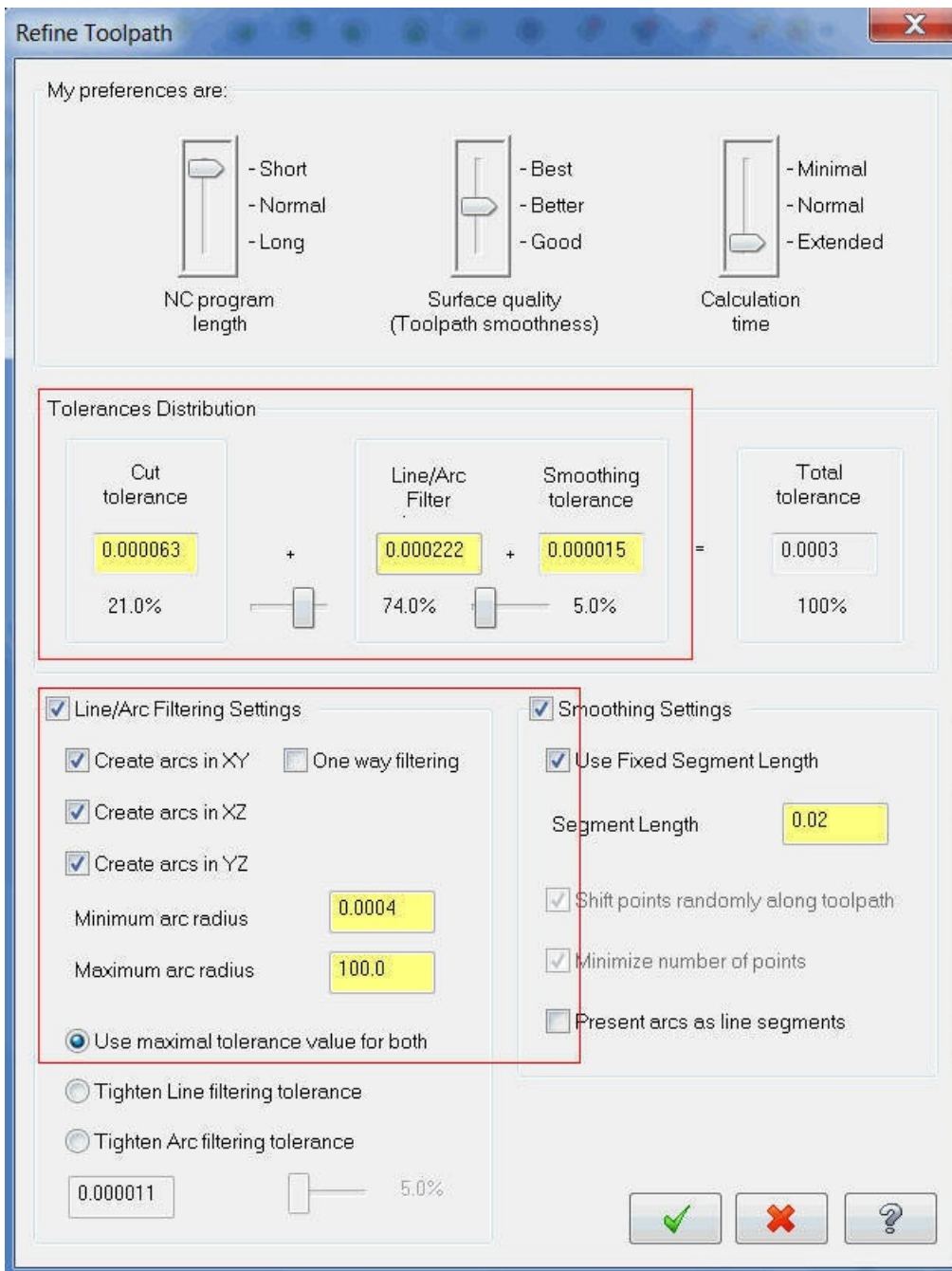
2

Most CAM systems have a 3D tool path refinement feature.

1 - Surface without tool path refinement.

2 - Surface with tool path refinement.

Filters and Tolerances



Mastercam's **Refine Toolpath** screen.

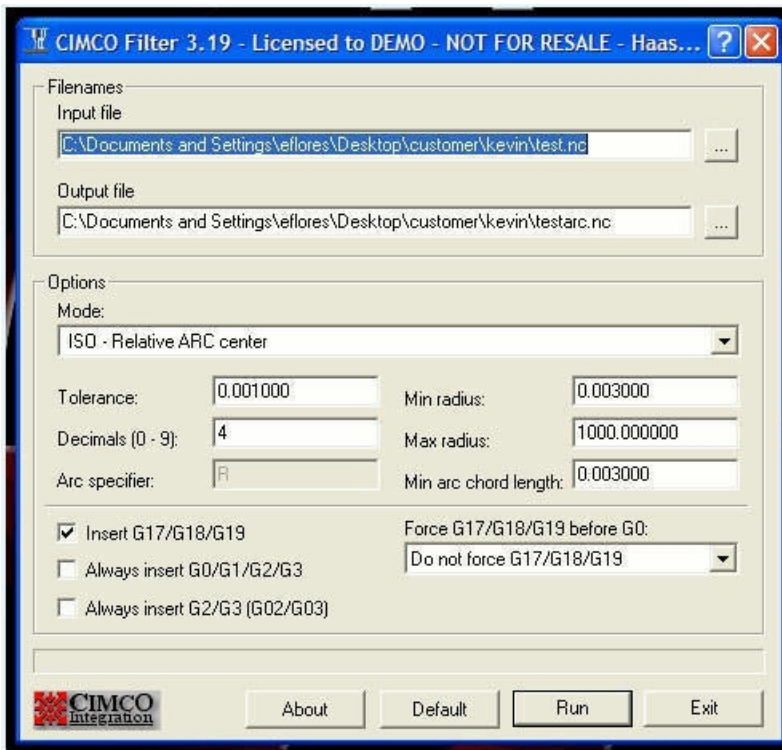
Depending on the type of toolpath or part complexity you may want to use MasterCam's Line/Arc Filtering Settings.

Set the Line/Arc filter to 25% and the Smoothing tolerance to 75%.

Mastercam features several different options to filter a particular tool path.

In general, try to set the cut tolerance between 0.0002" and 0.0005". This may be different on different types of materials and cutters.

This option allows you to create tight linear segments on a 3D model and then convert them to arc segments when you post the G-code. This results in smoother transitions and a better surface finish.



Third-party filters, like this software from Cimco, take the G-code created in your CAM system and convert the linear segments to arc segments. This produces smoother transitions and a better surface finish.

This software is useful for CAM systems that do not have an internal arc filter.

Machine Parameters

Setting **191**, **DEFAULT SMOOTHNESS** alters the feed acceleration parameters.

- Medium = default acceleration and corner rounding values.
- Rough = double the default acceleration and corner rounding values (shorter cycle times).
- Finish = half the default acceleration and corner rounding values (longer cycle time).

DEFAULT SMOOTHNESS Values can be overridden in the program with a G187 Px where:

- P1 = Rough
- P2 = Medium (default)
- P3 = Finish

Use G187 P1 to speed up roughing operations, leave stock, some corner rounding is possible.

Use G187 P3 to smooth out finishing operations.

COMMON	SOFTWARE	INPUT INVERT	SYSTEM	AXIS	COMPENSATION	DIAGNOSTICS			
X PARAM C									
345	X	SPINDLE THERM T. C.				0			
588	X	ENC. SCALE FACTOR				0			
620	X	PLUS TRAVEL LIMIT				0			
632	X	AXIS MOCON CHANNEL				100			
948	X	SECONDARY ENCODER C				0			
644	X	INDEXER INCREMENT				0			
656	X	INDEXER DWN TIMEOUT				0			
668	X	INDEXER DWN SETTLE				0			
680	X	LEAD COMPENS SHIFT				18			
762	X	AIR BRAKE ON DELAY				0			
774	X	ENCODER TYPE				7			
786	X	COMMAND FIR ORDER				64			
868	X	PHASE ADVANCE ANGLE				44			
884	X	NOTCH CENTER FREQ				230			
900	X	NOTCH DEPTH FACTOR				10			
916	X	MOCON ACC FFD COEF				0			
932	X	EREV PER MREV				5			
1016	X	DEADBAND				5			
1032	X	TORQUE CONSTANT				0			
1048	X	AMP PEAK CURRENT				0			
Parameter 786-797 COMMAND FIR FILTER ORDER									
X	Y	Z	A	B	Sp	U	V	W	TT

Parameter **786** , COMMAND FIR FILTER ORDER

The COMMAND FIR FILTER ORDER parameter for each axis provides smoothing at the start and finish of each acceleration/deceleration cycle.

If the machine has Sigma V motors, the FIR COMMAND ORDER paramter for each axis must be set to 64.


 **Caution:** Do not change the machine's parameters.

Notch Filter

Most machines have had their natural frequency mapped, and parameters established.

The notch filter monitors the machine's commanded motion.

If a machine is about to execute a move that is near its natural frequency, the notch filter will slightly alter the speed to avoid vibrations.

 **Caution:** Do not alter the machine's parameters.

Summary

Use the features in the CAM system to get the best surface finish.

Test different tool paths with different cut tolerance and smoothness settings in a block of material.

Record the techniques that work best for your part style and CAM system for future use.