MP21049 – How to Achieve Brilliant Surface Finishes for CNC Machining

Jeff Jaje and Wayne Griffenberg

Manufacturing Product Marketing Manager

Instagram & Twitter: JeffJaje

Class summary

Exploring the factors that affect surface finish. This is actually a very large topic, and the majority of time will be spent on the parameters in your CAM system and how they affect surface finish.

Key learning objectives

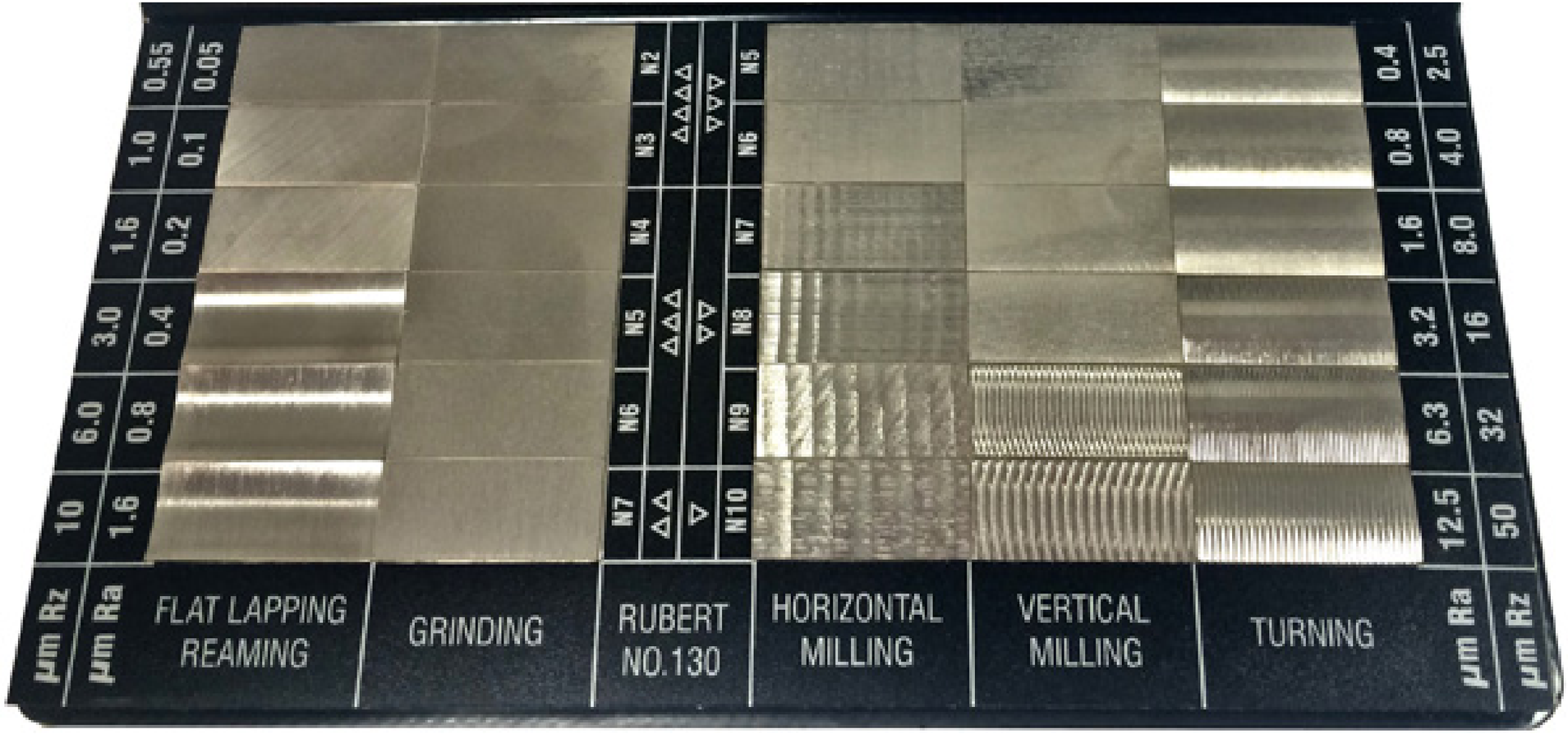
At the end of this class, you will be able to:

- Understand how tolerance affects surface finish
- Understand how stepover/cusp affects surface finish
- Understand how programming strategy affects surface finish
- See other aspects of the whole CNC milling workflow that affect surface finish

We will have time at the end of class for questions.

Surface Finish

Roughness, Waviness, and Measurement (µm or micron):



Surface Finish

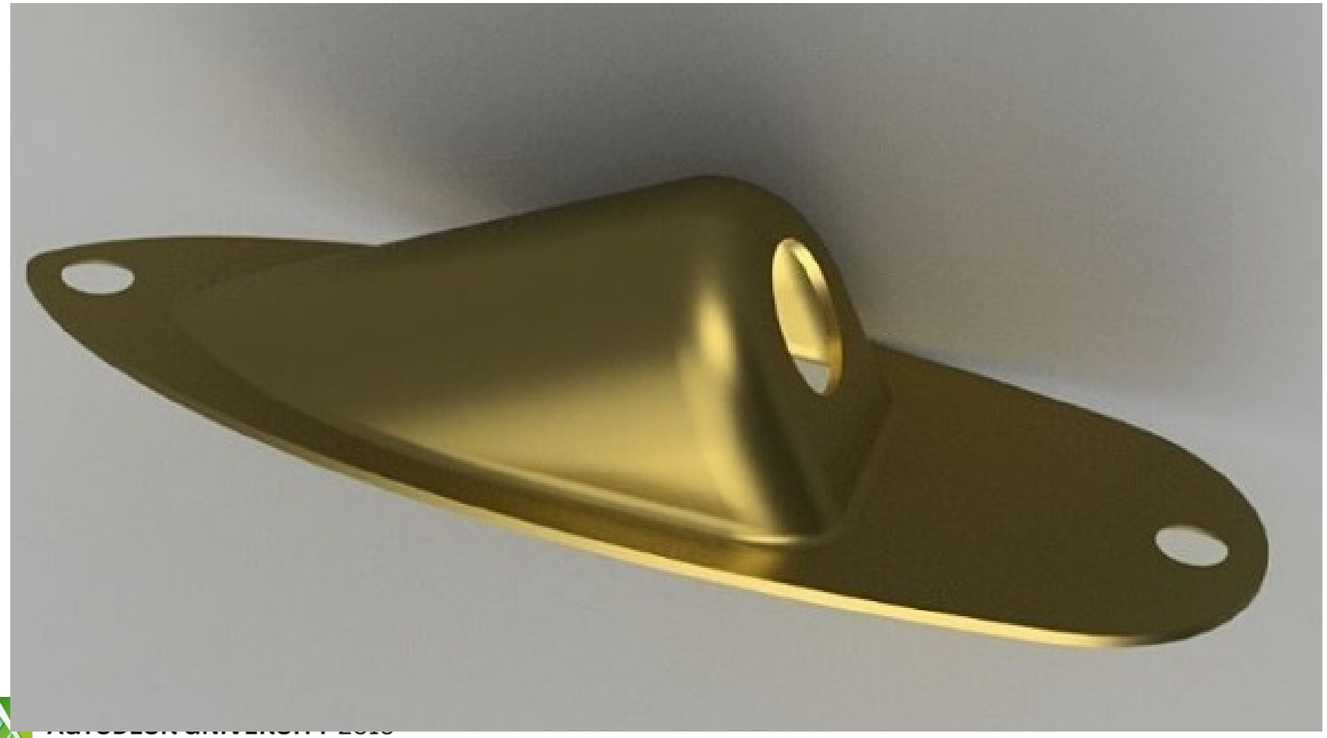
- Roughness – finely spaced surface irregularities. In engineering, what is usually meant by surface finish.
- Waviness – measure of surface irregularities larger than roughness; usually from deflection, warping, vibrations.
- Measurement: actual amount
  - Contact – Stylus
  - Non Contact – interferometry, microscopy, structured light, etc.

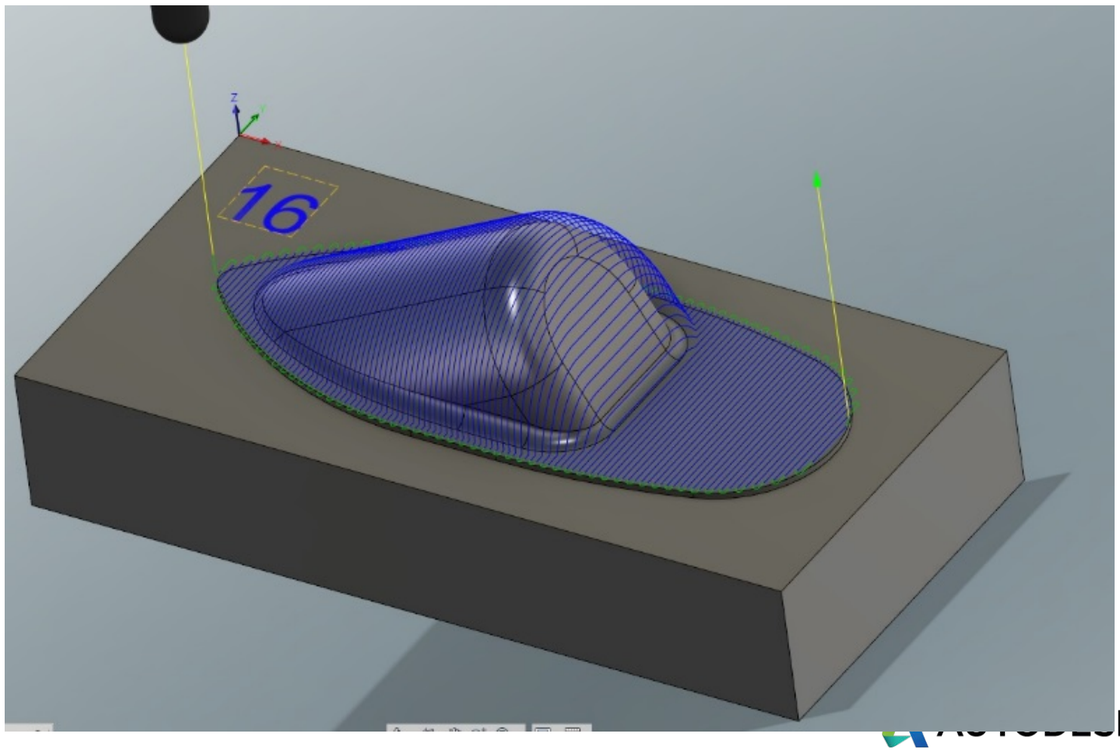
Street Definition

- How much time will they have to spend polishing the parts?

Part

Wanted to test on something small, with curvature, steep areas, and radii.

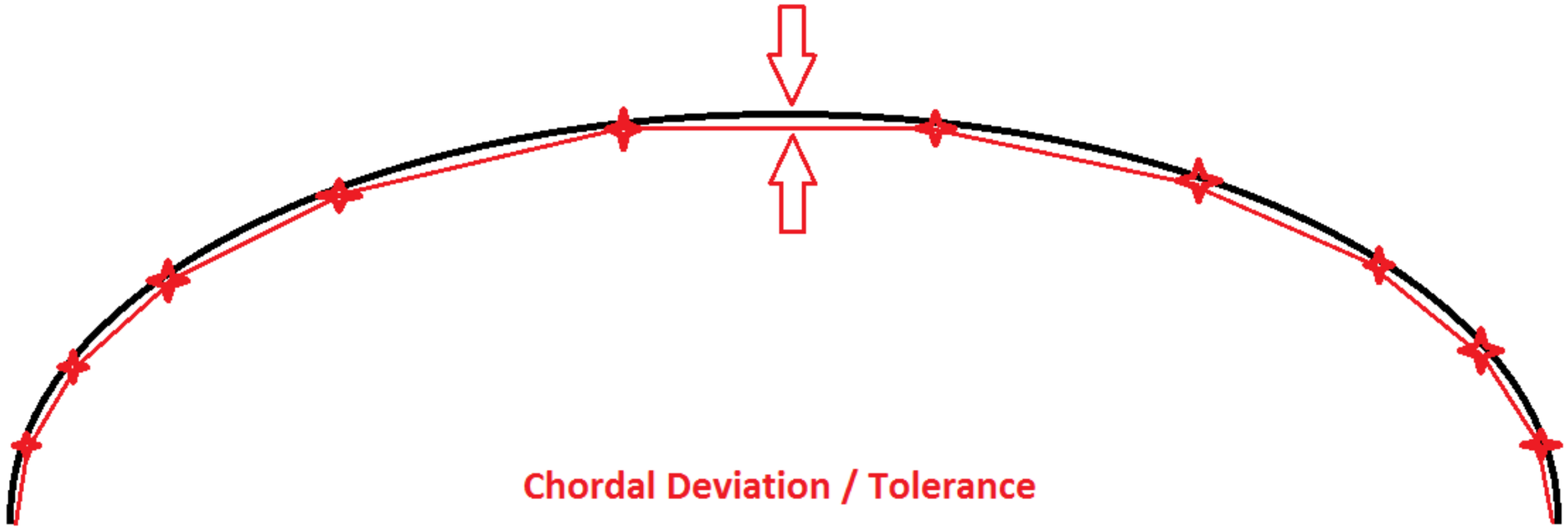




Tolerance

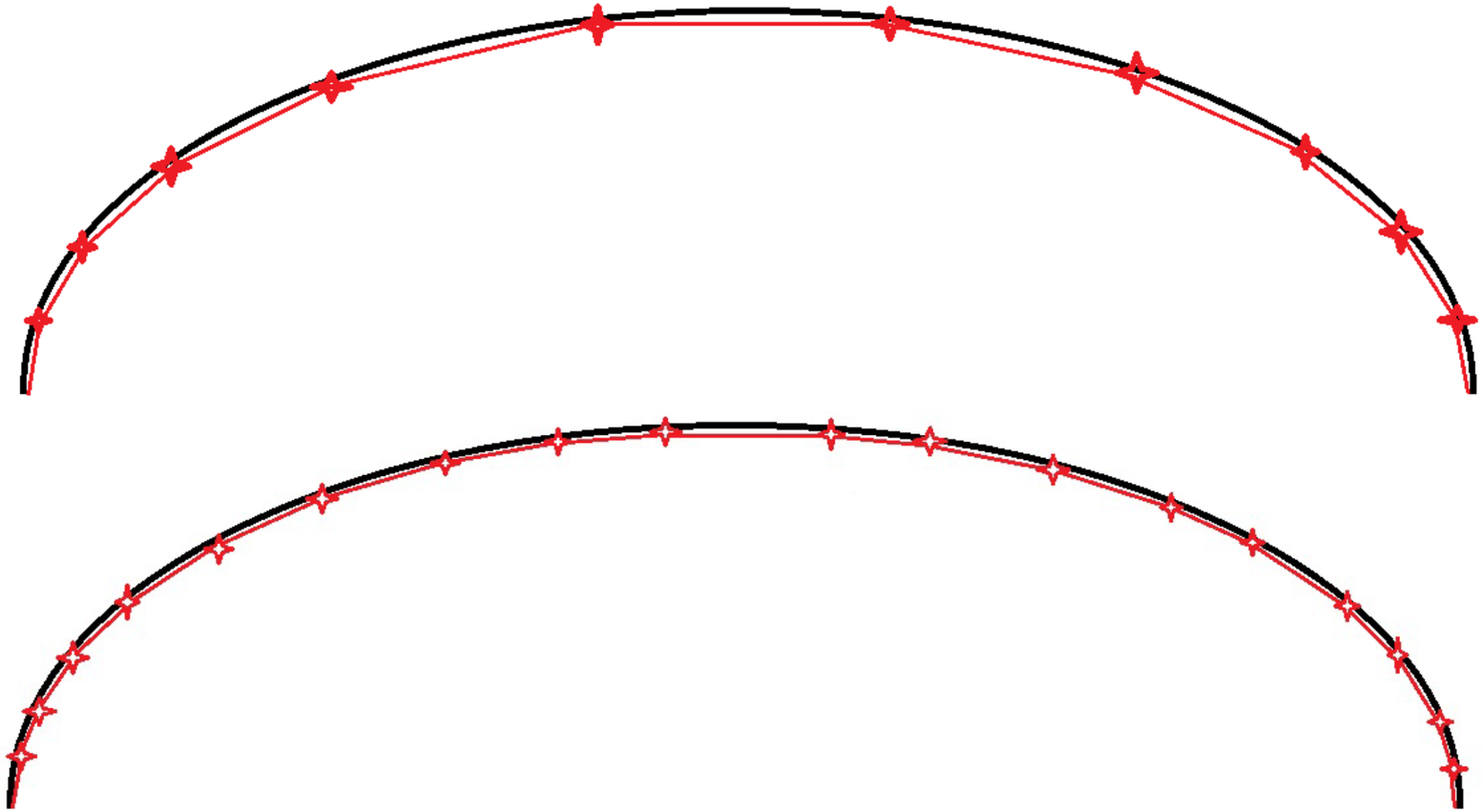
Tolerance – Chordal Deviation

The distance from the point to point movements to the original CAD curve or surface:



Tolerance – tighter tolerance = more points

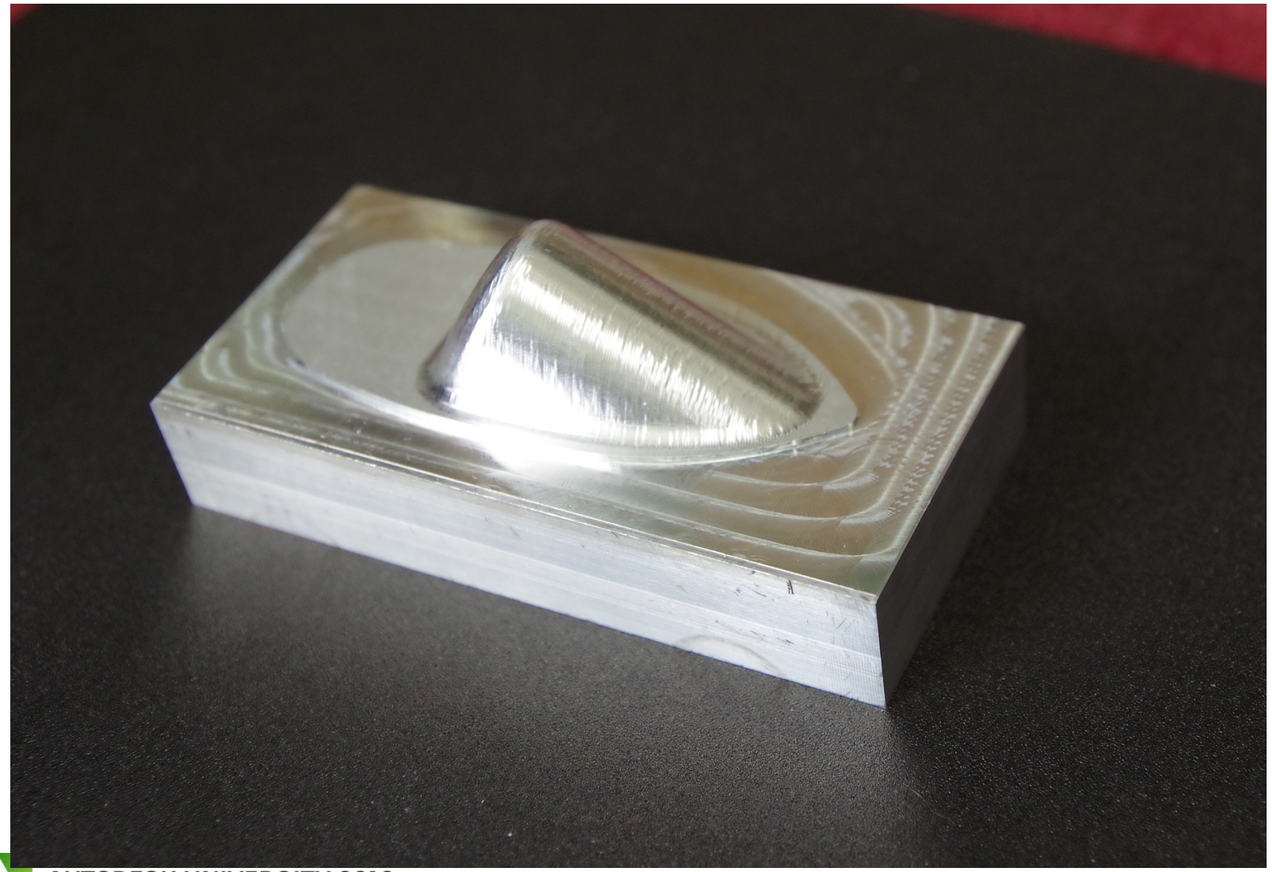
The tolerance you use when programming is the first important parameter to quality finish:



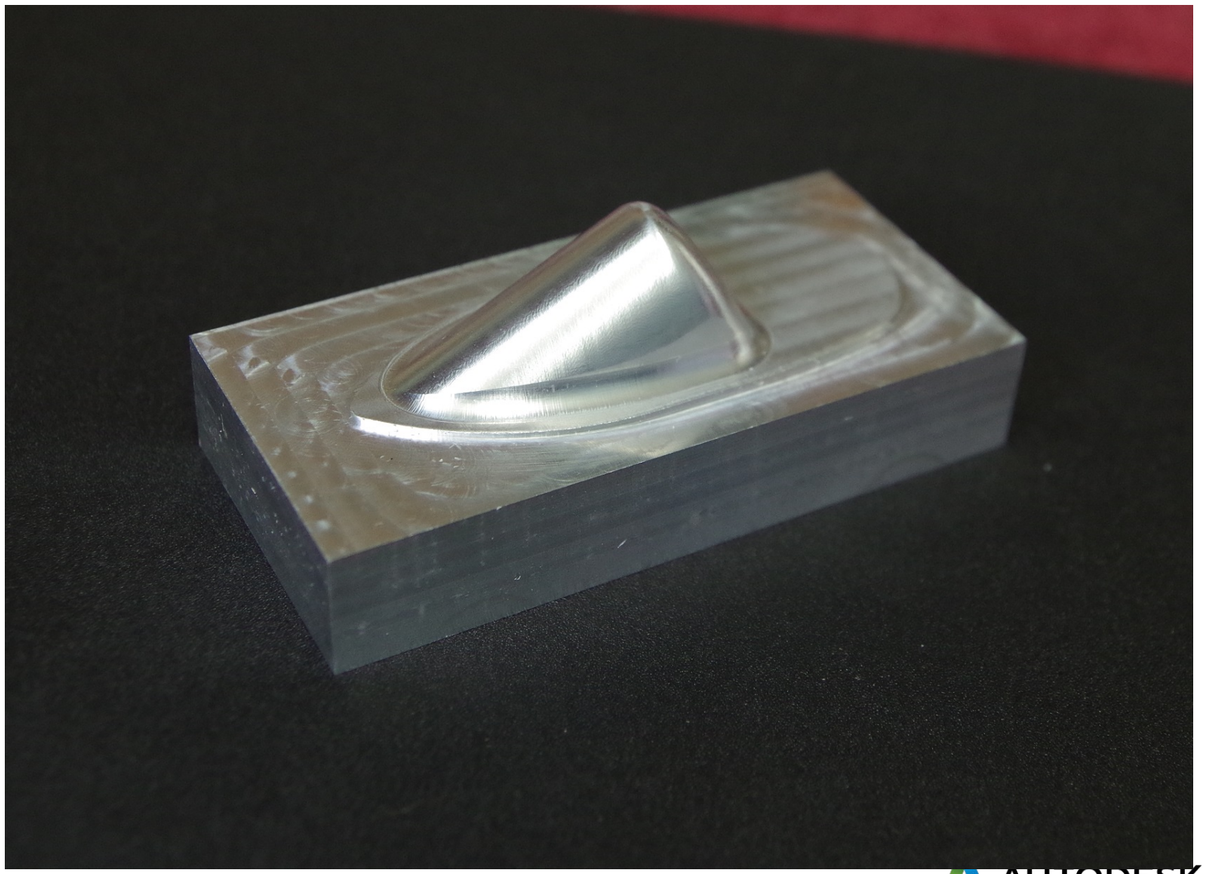
Tolerance

The tolerance you use when programming is the first important parameter to quality finish:

#4 0.12 mm tolerance

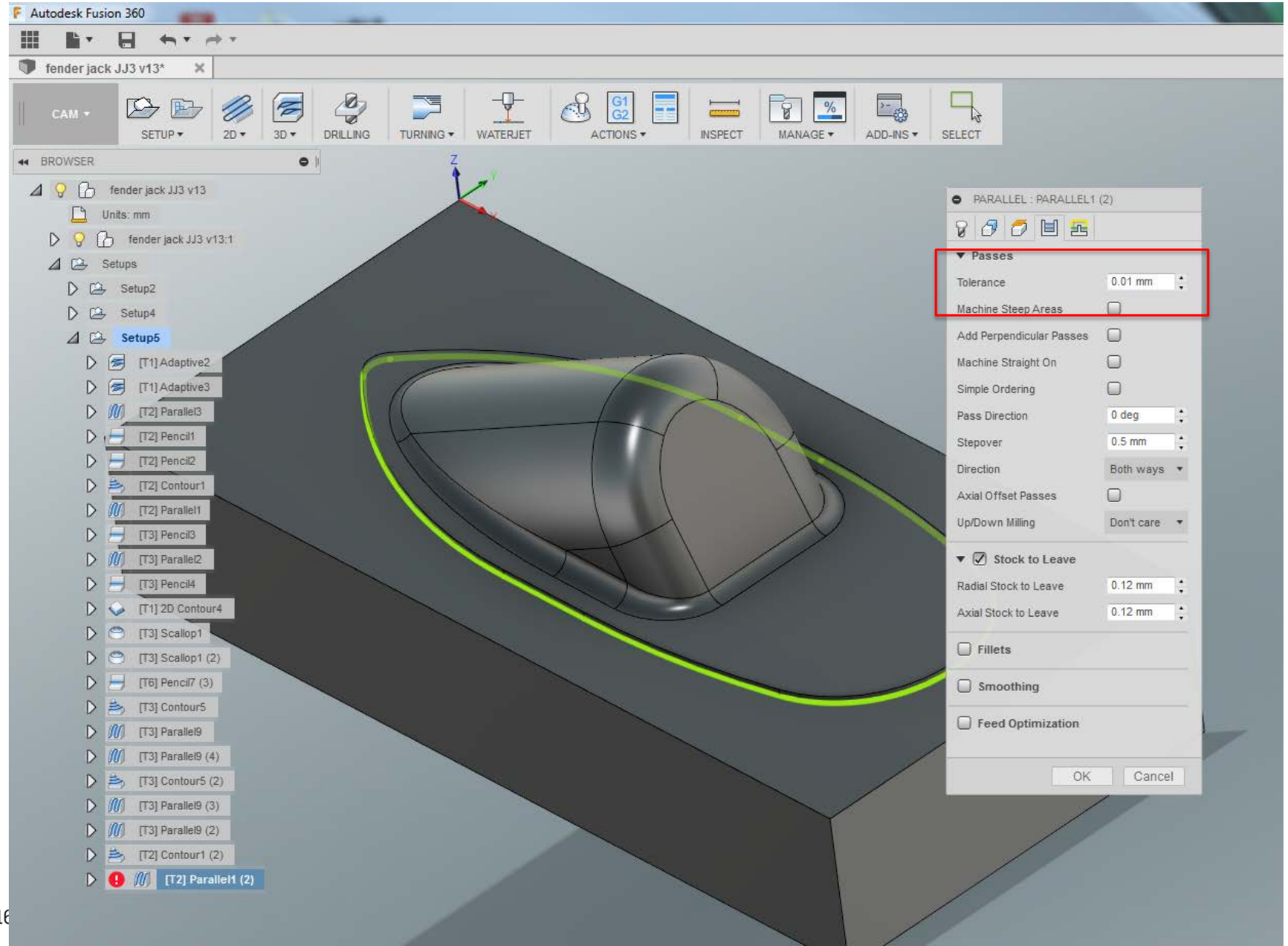


#5 0.001 mm tolerance

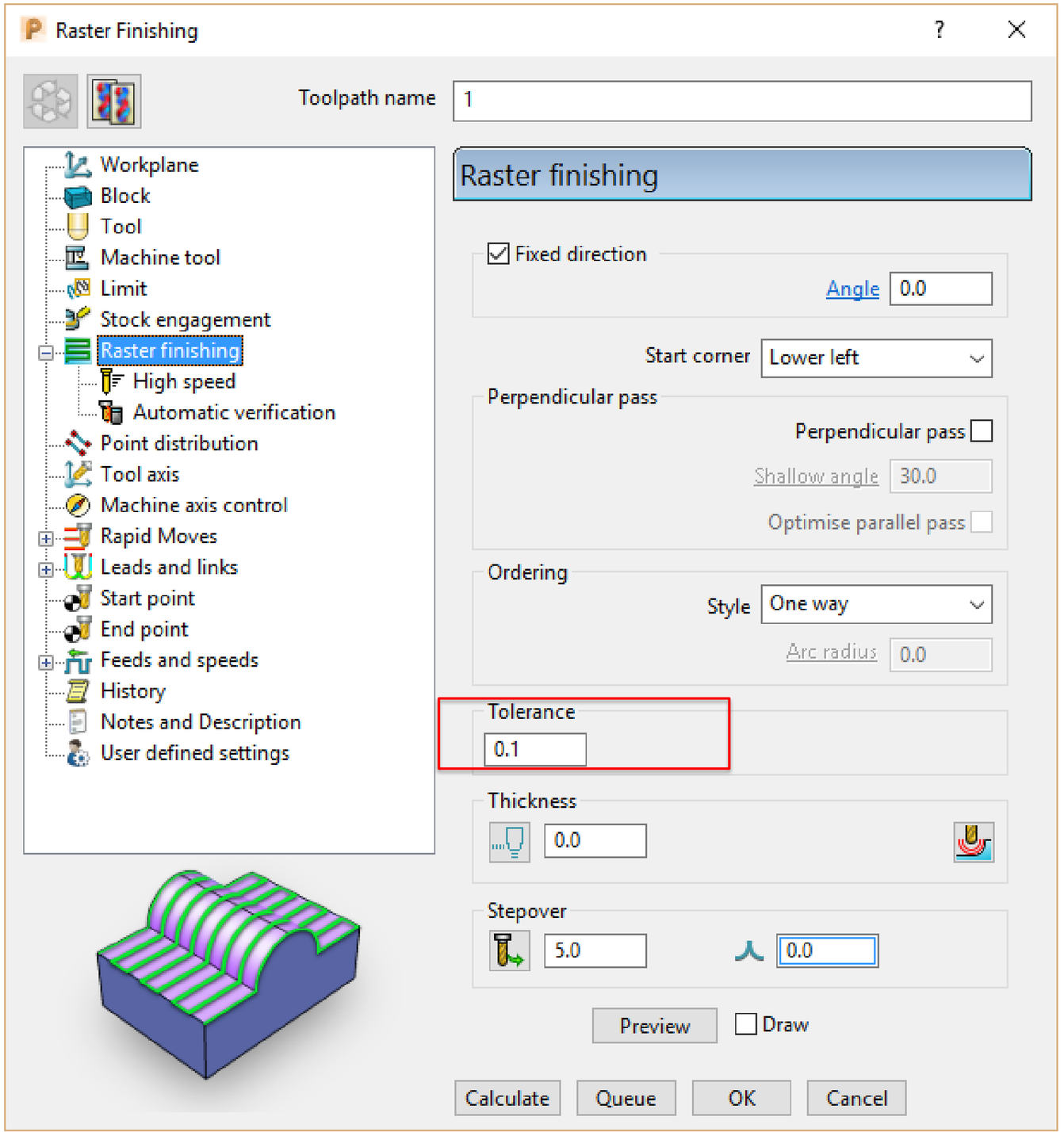


Also #3 0.01mm tolerance

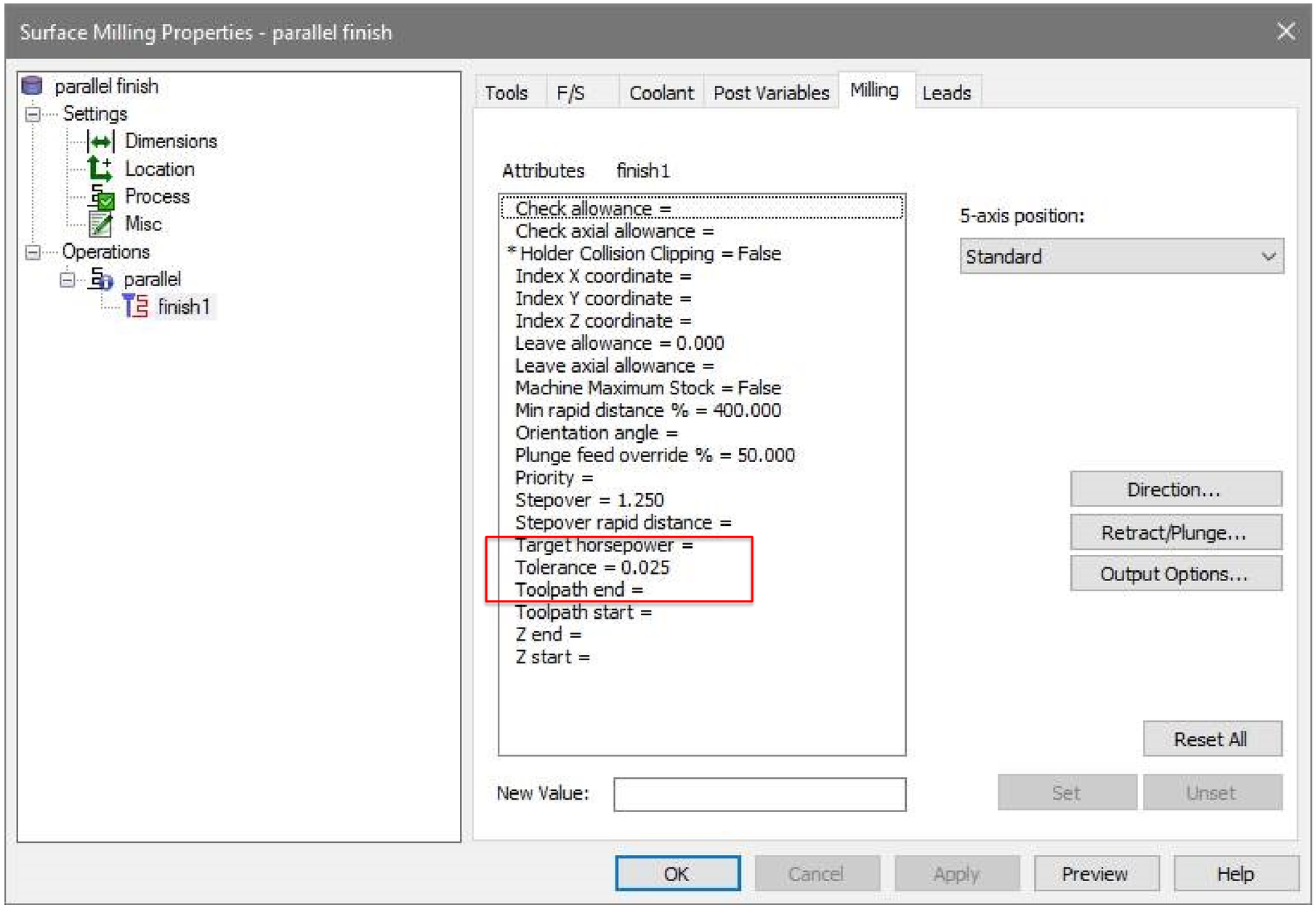
Fusion 360


Autodesk Fusion 360
fender jack JJ3 v13*
CAM
SETUP
2D
3D
DRILLING
TURNING
WATERJET
ACTIONS
INSPECT
MANAGE
ADD-INS
SELECT
BROWSER
fender jack JJ3 v13
Units: mm
fender jack JJ3 v13:1
Setups
Setup2
Setup4
Setup5
[T1] Adaptive2
[T1] Adaptive3
[T2] Parallel3
[T2] Pencil1
[T2] Pencil2
[T2] Contour1
[T2] Parallel1
[T3] Pencil3
[T3] Parallel2
[T3] Pencil4
[T1] 2D Contour4
[T3] Scallop1
[T3] Scallop1 (2)
[T6] Pencil7 (3)
[T3] Contour5
[T3] Parallel9
[T3] Parallel9 (4)
[T3] Contour5 (2)
[T3] Parallel9 (3)
[T3] Parallel9 (2)
[T2] Contour1 (2)
[T2] Parallel1 (2)
PARALLEL : PARALLEL1 (2)
Passes
Tolerance
0.01 mm
Machine Steep Areas
Add Perpendicular Passes
Machine Straight On
Simple Ordering
Pass Direction
0 deg
Stepover
0.5 mm
Direction
Both ways
Axial Offset Passes
Up/Down Milling
Don't care
Stock to Leave
Radial Stock to Leave
0.12 mm
Axial Stock to Leave
0.12 mm
Fillets
Smoothing
Feed Optimization
OK
Cancel

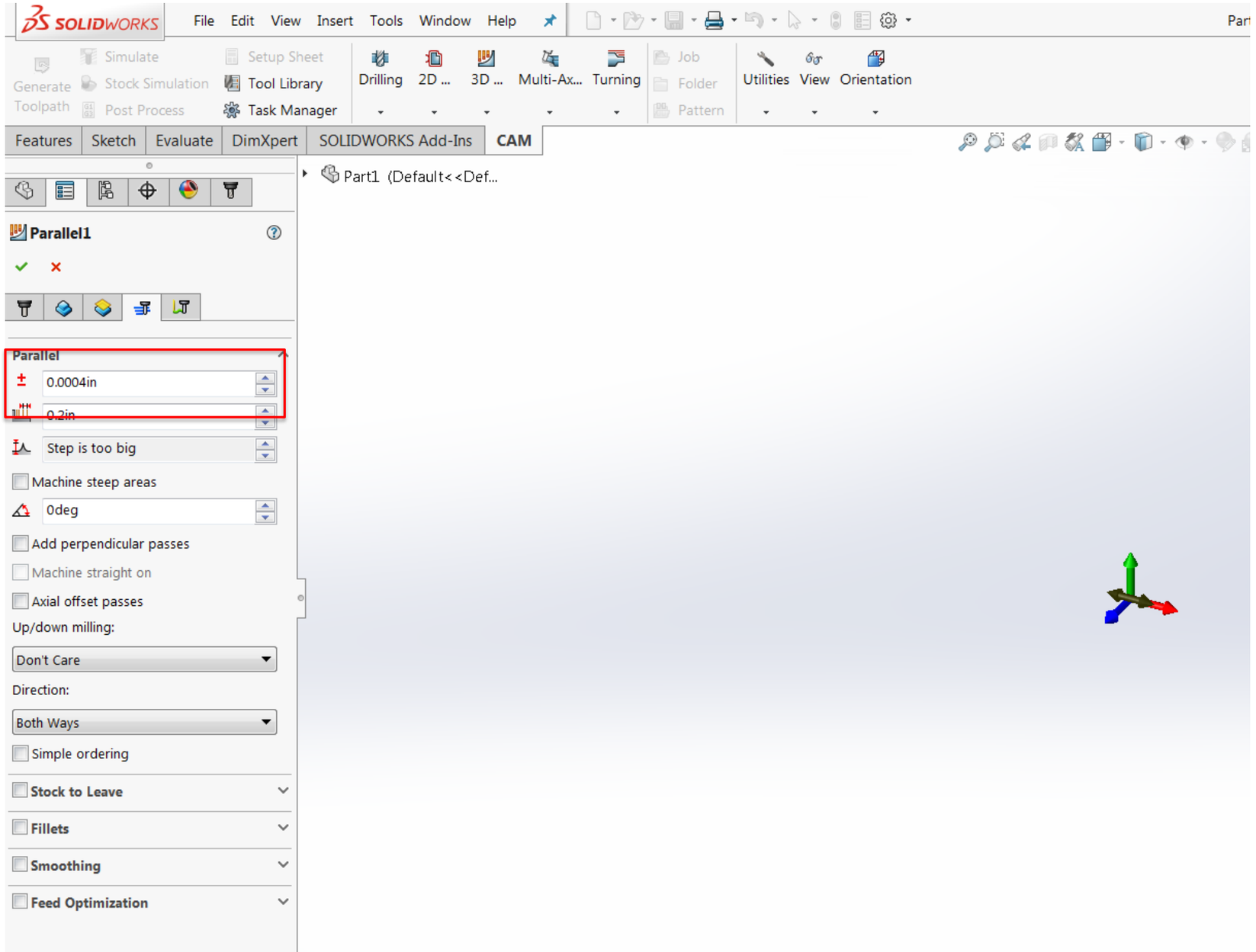
PowerMill


Raster Finishing
Toolpath name
1
Workplane
Block
Tool
Machine tool
Limit
Stock engagement
Raster finishing
High speed
Automatic verification
Point distribution
Tool axis
Machine axis control
Rapid Moves
Leads and links
Start point
End point
Feeds and speeds
History
Notes and Description
User defined settings
Raster finishing
Fixed direction
Angle
0.0
Start corner
Lower left
Perpendicular pass
Perpendicular pass
Shallow angle
30.0
Optimise parallel pass
Ordering
Style
One way
Arc radius
0.0
Tolerance
0.1
Thickness
0.0
Stepover
5.0
0.0
Preview
Draw
Calculate
Queue
OK
Cancel

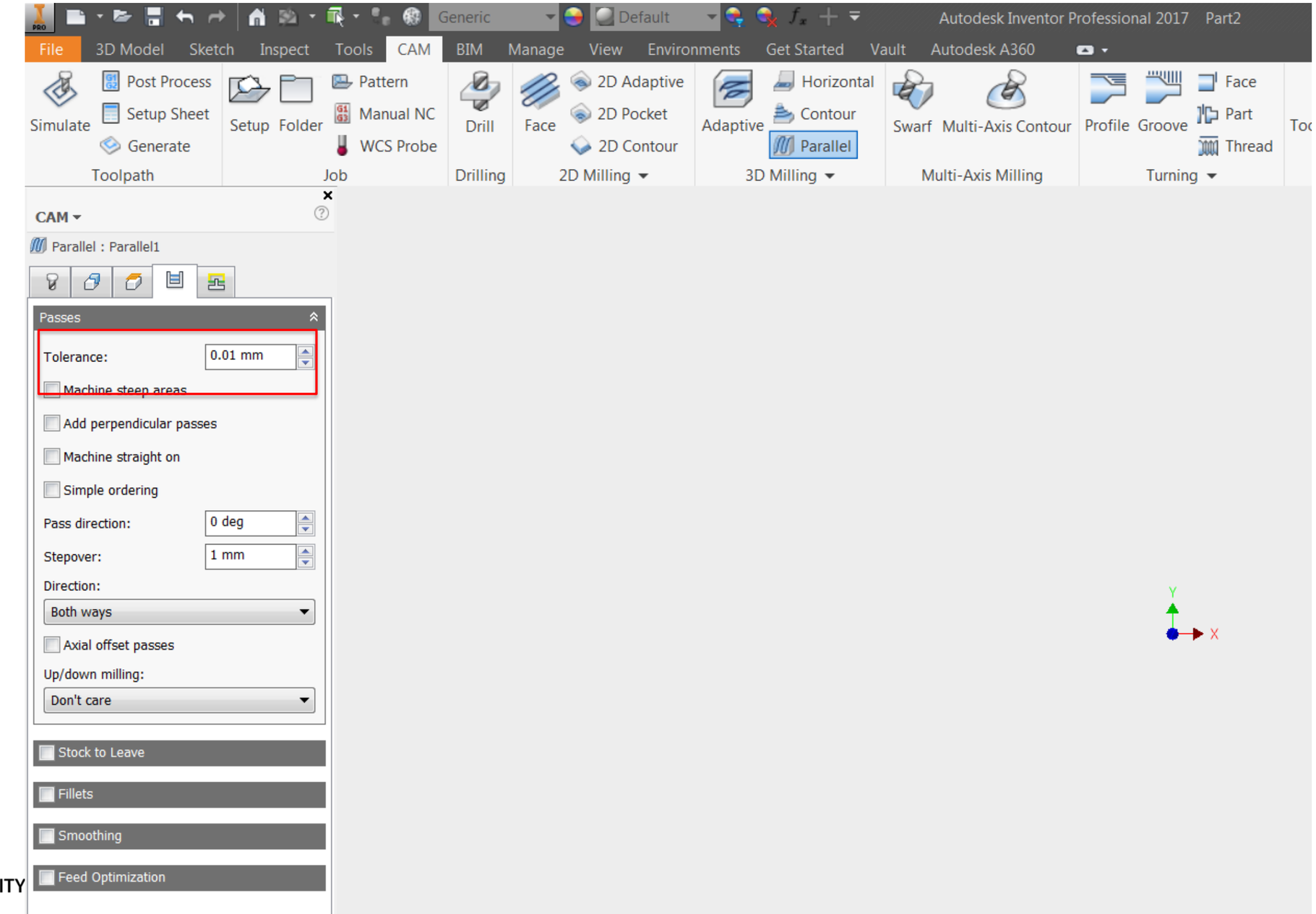
FeatureCAM


Surface Milling Properties - parallel finish
parallel finish
Settings
Dimensions
Location
Process
Misc
Operations
parallel
finish1
Tools
F/S
Coolant
Post Variables
Milling
Leads
Attributes
finish1
Check allowance =
Check axial allowance =
* Holder Collision Clipping = False
Index X coordinate =
Index Y coordinate =
Index Z coordinate =
Leave allowance = 0.000
Leave axial allowance =
Machine Maximum Stock = False
Min rapid distance % = 400.000
Orientation angle =
Plunge feed override % = 50.000
Priority =
Stepover = 1.250
Stepover rapid distance =
Target horsepower =
Tolerance = 0.025
Toolpath end =
Toolpath start =
Z end =
Z start =
5-axis position:
Standard
Direction...
Retract/Plunge...
Output Options...
Reset All
New Value:
Set
Unset
OK
Cancel
Apply
Preview
Help

HSMWorks


SOLIDWORKS
File Edit View Insert Tools Window Help
Simulate
Stock Simulation
Post Process
Setup Sheet
Tool Library
Task Manager
Generate Toolpath
Drilling
2D ...
3D ...
Multi-Ax...
Turning
Job
Folder
Pattern
Utilities
View
Orientation
Features Sketch Evaluate DimXpert SOLIDWORKS Add-Ins CAM
Part1 (Default<<Def...
Parallel1
Parallel
0.0004in
0.2in
Step is too big
Machine steep areas
0deg
Add perpendicular passes
Machine straight on
Axial offset passes
Up/down milling:
Don't Care
Direction:
Both Ways
Simple ordering
Stock to Leave
Fillets
Smoothing
Feed Optimization

Inventor HSM



Tolerance: Mill

Not only do you have the tolerance of the CAM software but you also have the tolerance of the mill and controller.

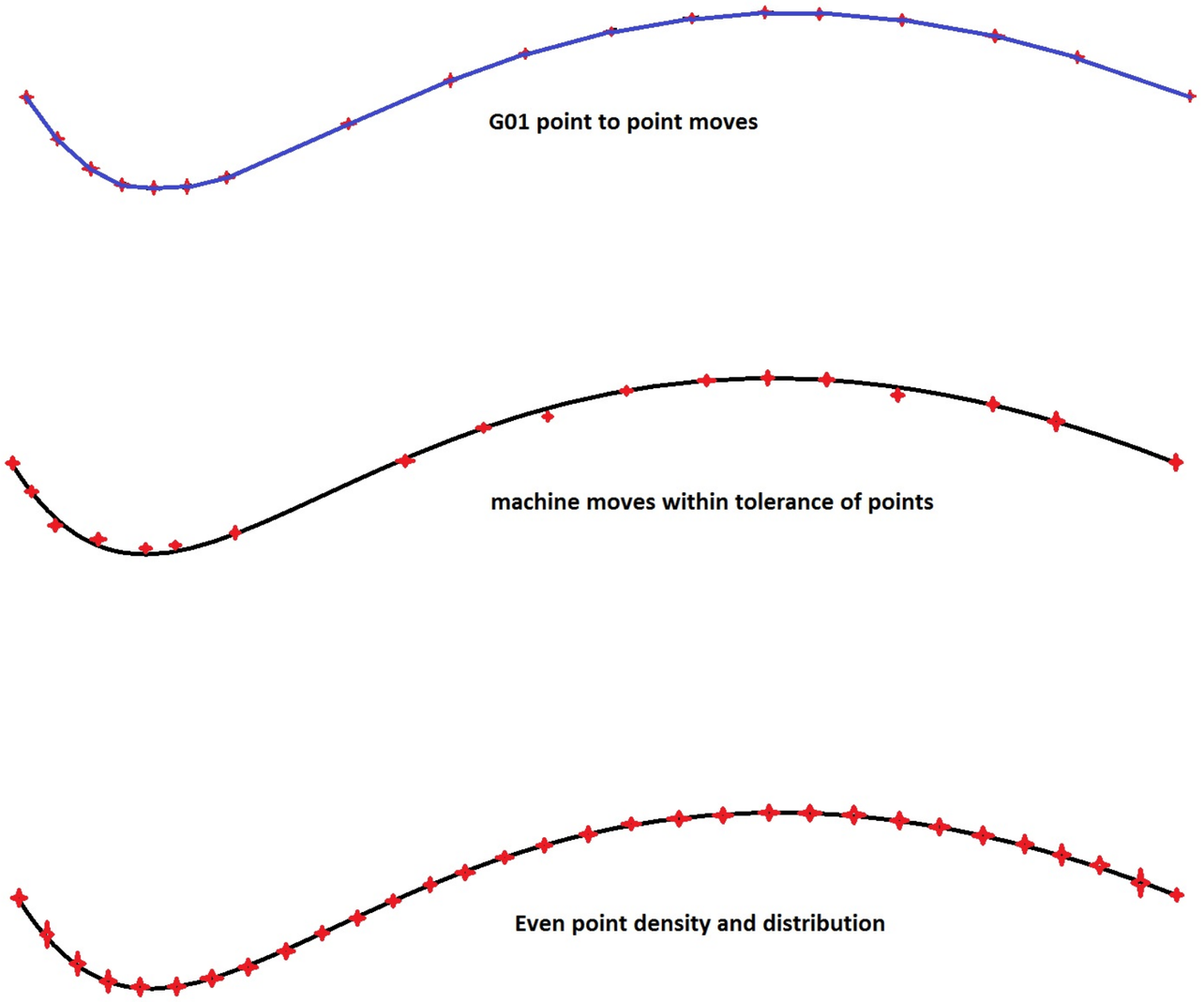
Tolerance: Mill

Do CNC mills go exactly through every point?

No most have their own tolerances too!

Point Density – How many points (tolerance)

Point Distribution – Even is better



Tolerance: Mill (smoothness while running too)

Some machines:

- Fanuc: P10000 (and others)
- Makino: M250, M251, M252
- Heidenhain: CYCLE DEF 32.0 Tolerance
- Haas: G187 Px E.xx

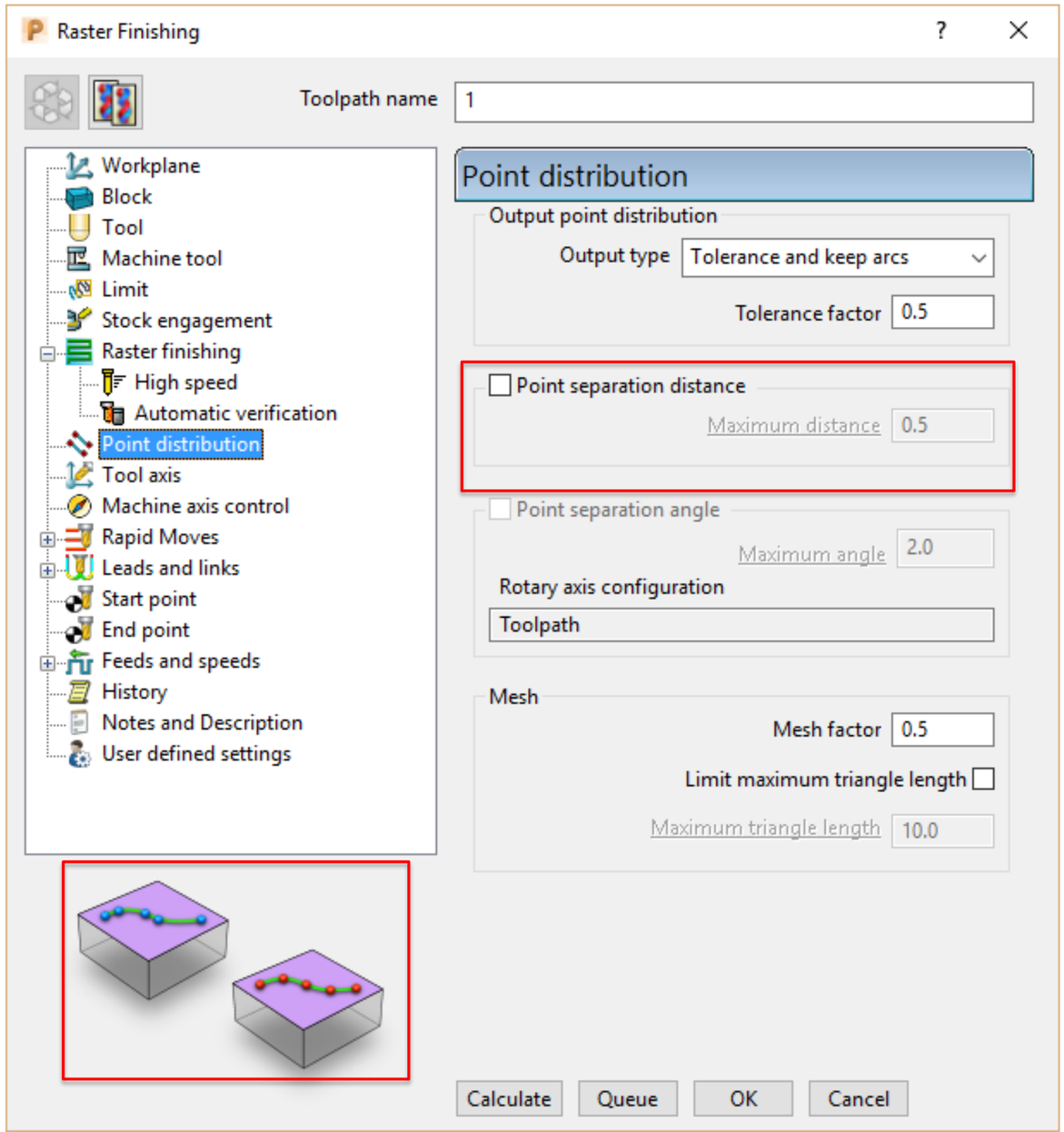
Tolerance: Mill

If your machine tolerance is low, say .001” – is there any benefit to programming the CAM tolerance tighter?

It depends:

- More points may still help out the controllers splining algorithm
- Too many points may overload the controller.
- Super dense points may introduce “noise” variations making for a slow cut.

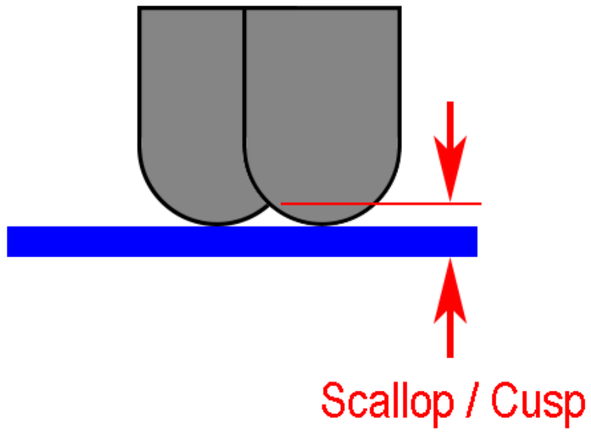
PowerMill Parameters – Point Distribution

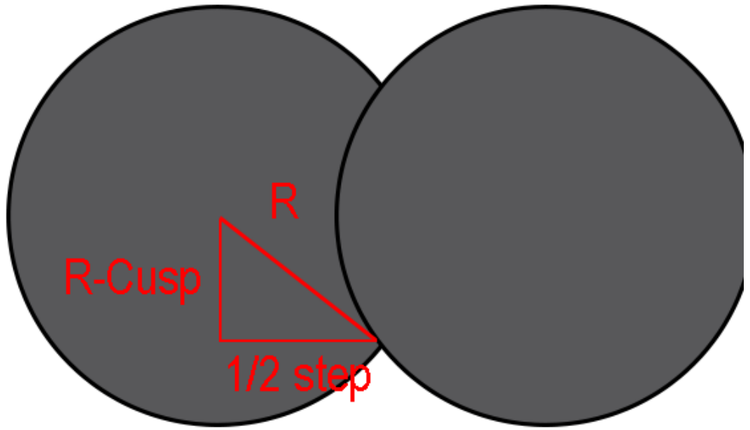


Stepover/Stepdown – cusp/scallop

Cusp Height

Peak Height from stepover distance

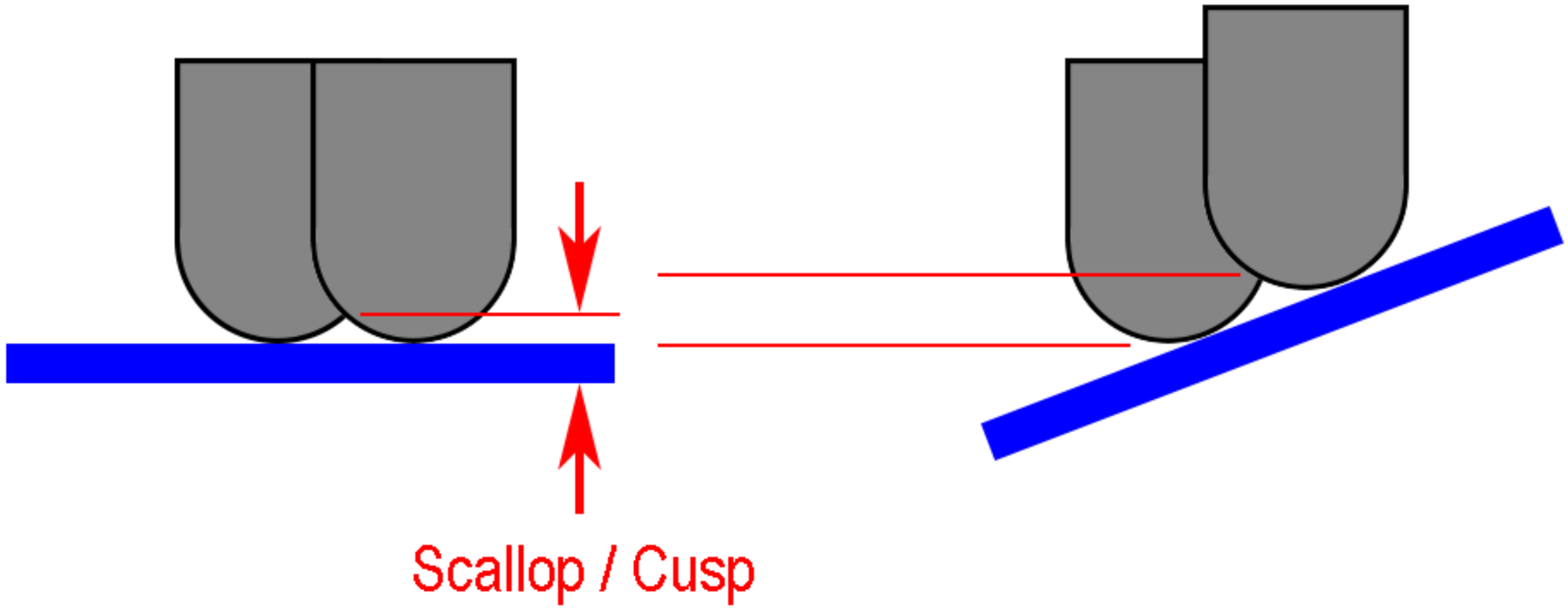




Step= 2 * SQRT [R^2-(R-cusp)^2]

Cusp Height

Peak Height from stepover distance



Scallop and Stepover

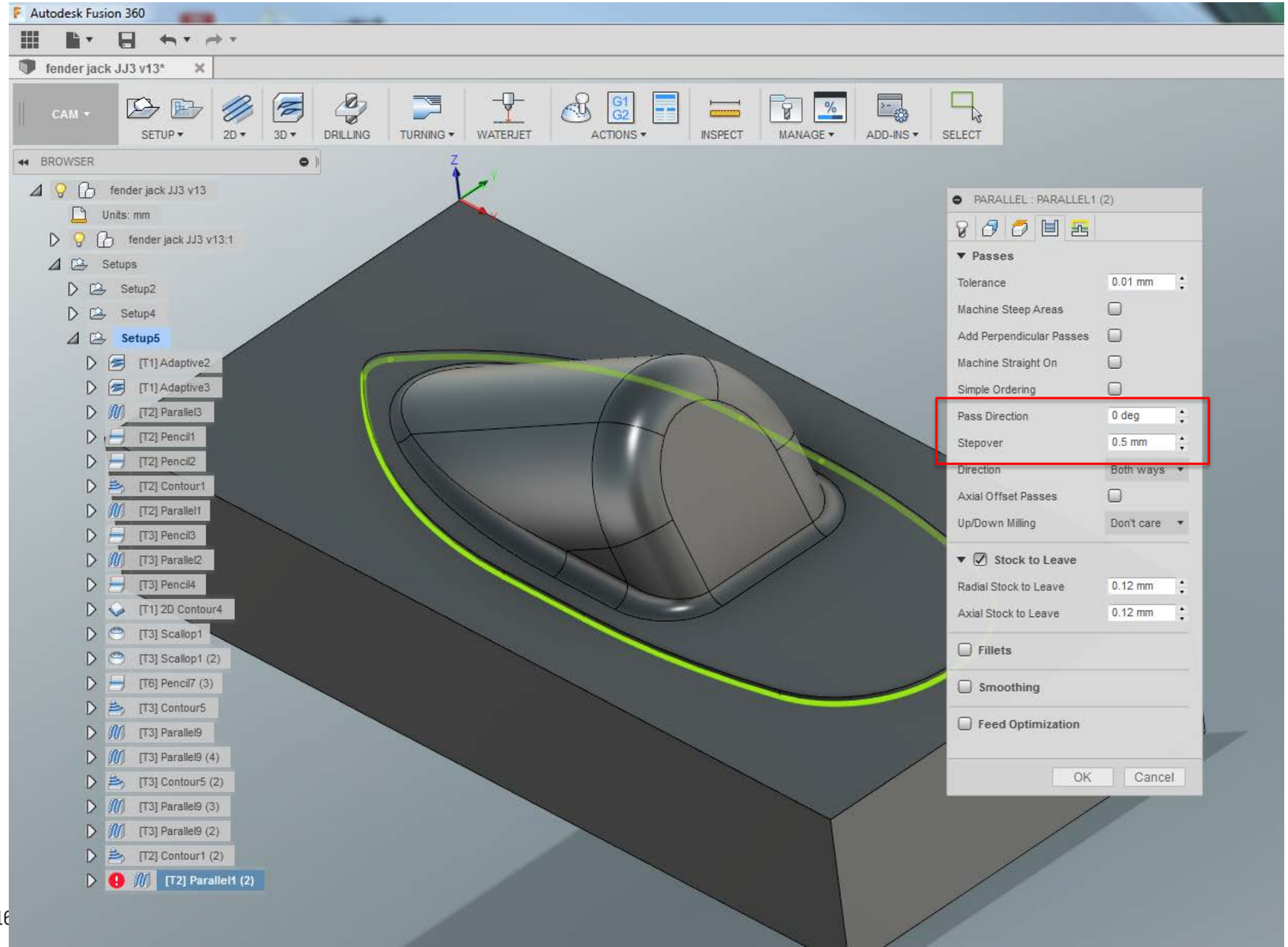
The traditional surface finish provided mainly by your stepover “Roughness”

#1 0.8 step 0.025 scallop

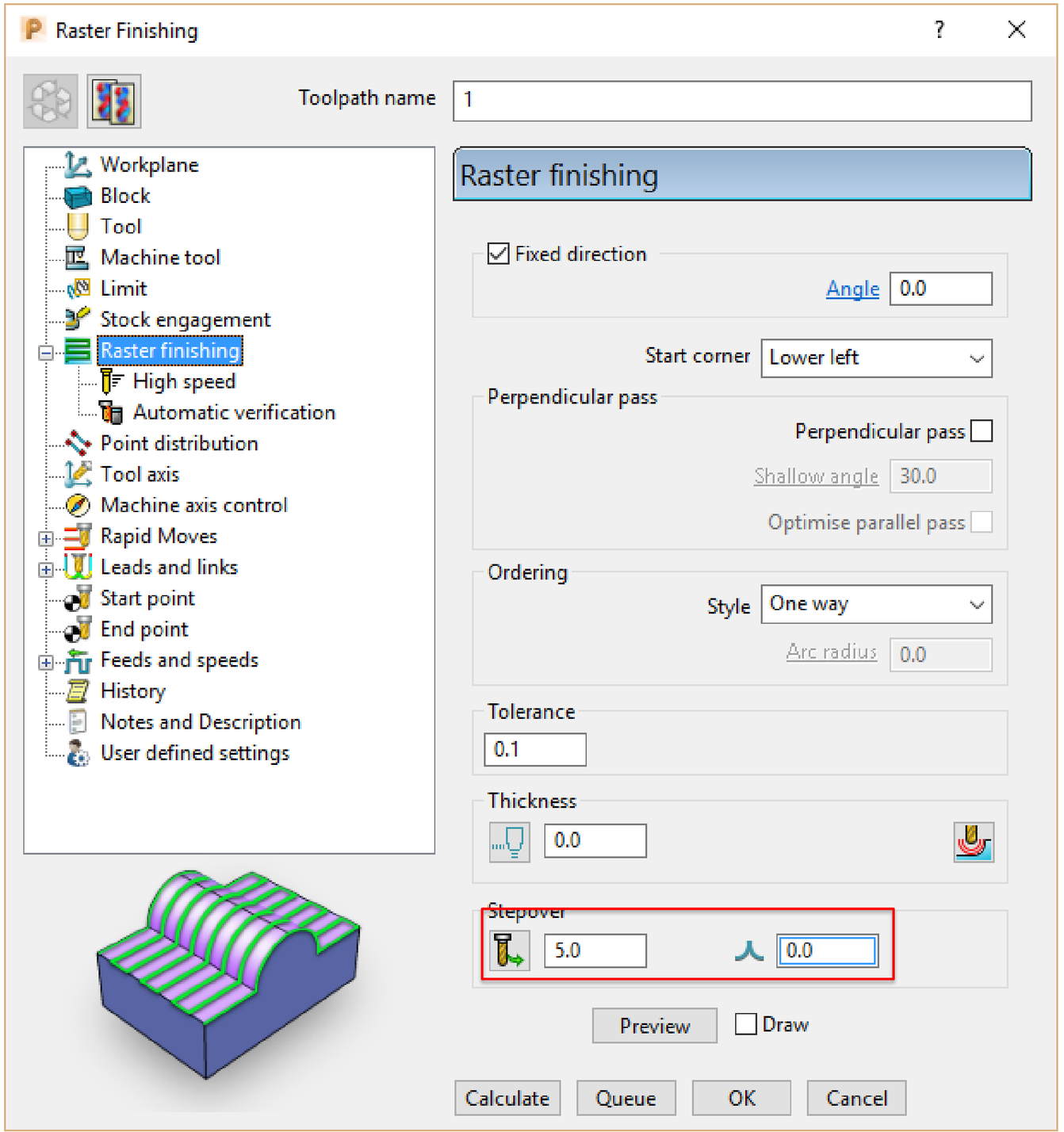
#2 0.25 step 0.0025 scallop

Also #3 (.16mm step)

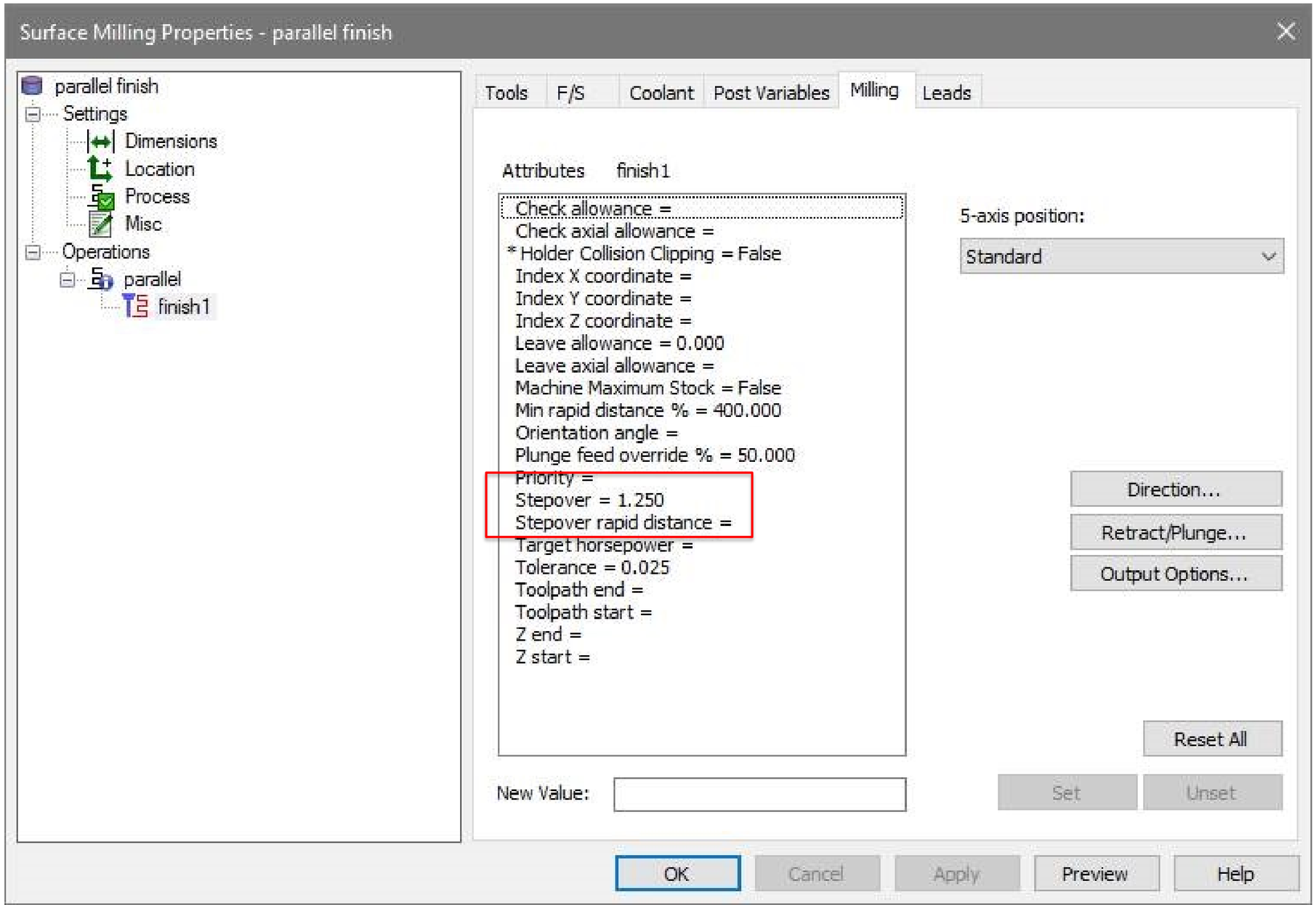
Fusion 360


Autodesk Fusion 360
fender jack JJ3 v13*
CAM
SETUP
2D
3D
DRILLING
TURNING
WATERJET
ACTIONS
INSPECT
MANAGE
ADD-INS
SELECT
BROWSER
fender jack JJ3 v13
Units: mm
fender jack JJ3 v13:1
Setups
Setup2
Setup4
Setup5
[T1] Adaptive2
[T1] Adaptive3
[T2] Parallel3
[T2] Pencil1
[T2] Pencil2
[T2] Contour1
[T2] Parallel1
[T3] Pencil3
[T3] Parallel2
[T3] Pencil4
[T1] 2D Contour4
[T3] Scallop1
[T3] Scallop1 (2)
[T6] Pencil7 (3)
[T3] Contour5
[T3] Parallel9
[T3] Parallel9 (4)
[T3] Contour5 (2)
[T3] Parallel9 (3)
[T3] Parallel9 (2)
[T2] Contour1 (2)
[T2] Parallel1 (2)
PARALLEL : PARALLEL1 (2)
Passes
Tolerance
0.01 mm
Machine Steep Areas
Add Perpendicular Passes
Machine Straight On
Simple Ordering
Pass Direction
0 deg
Stepover
0.5 mm
Direction
Both ways
Axial Offset Passes
Up/Down Milling
Don't care
Stock to Leave
Radial Stock to Leave
0.12 mm
Axial Stock to Leave
0.12 mm
Fillets
Smoothing
Feed Optimization
OK
Cancel

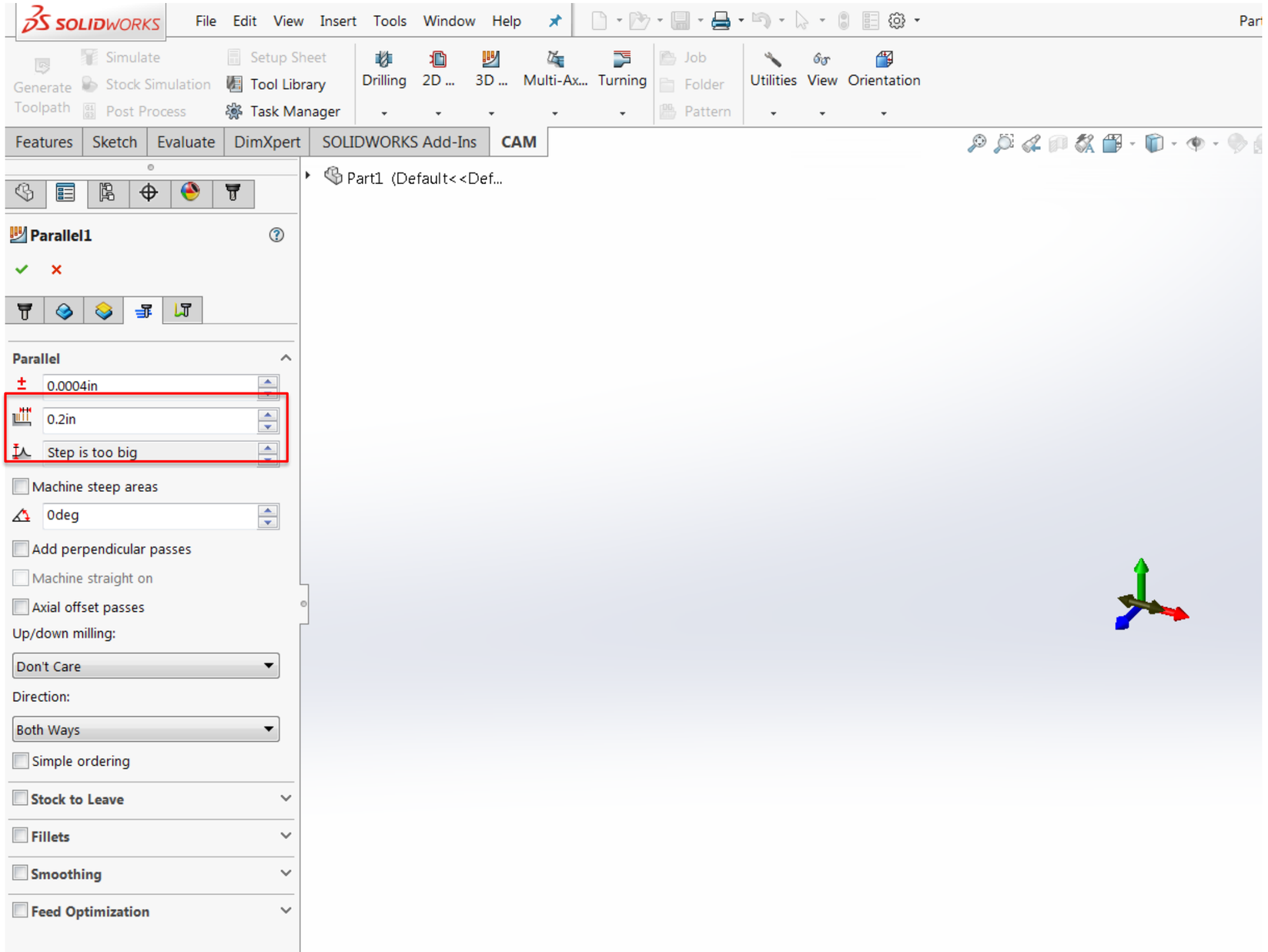
PowerMill


Raster Finishing
Toolpath name
1
Workplane
Block
Tool
Machine tool
Limit
Stock engagement
Raster finishing
High speed
Automatic verification
Point distribution
Tool axis
Machine axis control
Rapid Moves
Leads and links
Start point
End point
Feeds and speeds
History
Notes and Description
User defined settings
Raster finishing
Fixed direction
Angle
0.0
Start corner
Lower left
Perpendicular pass
Perpendicular pass
Shallow angle
30.0
Optimise parallel pass
Ordering
Style
One way
Arc radius
0.0
Tolerance
0.1
Thickness
0.0
Stepover
5.0
0.0
Preview
Draw
Calculate
Queue
OK
Cancel

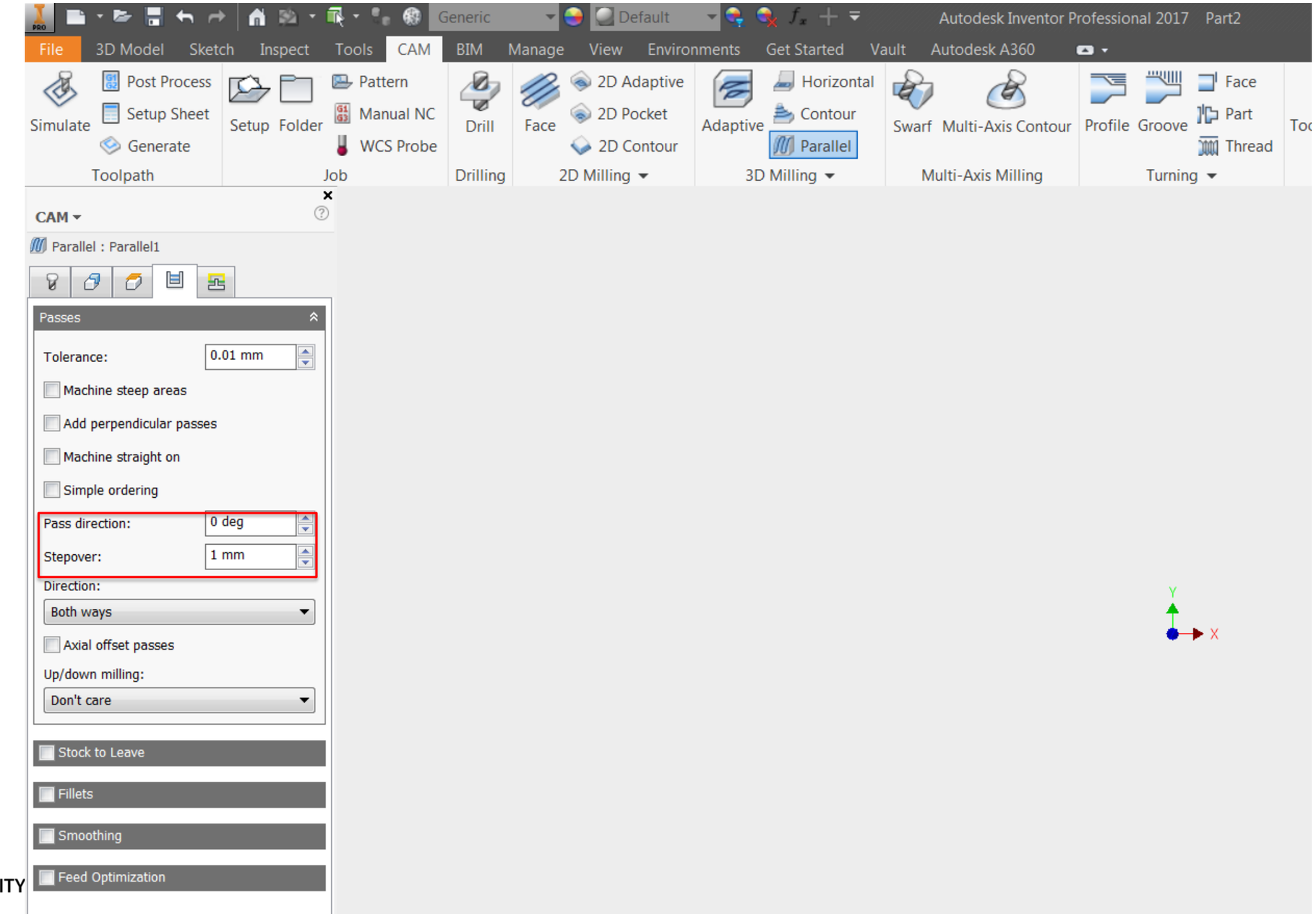
FeatureCAM


Surface Milling Properties - parallel finish
parallel finish
Settings
Dimensions
Location
Process
Misc
Operations
parallel
finish1
Tools
F/S
Coolant
Post Variables
Milling
Leads
Attributes
finish1
Check allowance =
Check axial allowance =
* Holder Collision Clipping = False
Index X coordinate =
Index Y coordinate =
Index Z coordinate =
Leave allowance = 0.000
Leave axial allowance =
Machine Maximum Stock = False
Min rapid distance % = 400.000
Orientation angle =
Plunge feed override % = 50.000
Priority =
Stepover = 1.250
Stepover rapid distance =
Target horsepower =
Tolerance = 0.025
Toolpath end =
Toolpath start =
Z end =
Z start =
5-axis position:
Standard
Direction...
Retract/Plunge...
Output Options...
Reset All
New Value:
Set
Unset
OK
Cancel
Apply
Preview
Help

HSMWorks


SOLIDWORKS
File Edit View Insert Tools Window Help
Simulate
Stock Simulation
Post Process
Setup Sheet
Tool Library
Task Manager
Generate Toolpath
Drilling
2D ...
3D ...
Multi-Ax...
Turning
Job
Folder
Pattern
Utilities
View
Orientation
Features Sketch Evaluate DimXpert SOLIDWORKS Add-Ins CAM
Part1 (Default<<Def...
Parallel1
Parallel
0.0004in
0.2in
Step is too big
Machine steep areas
0deg
Add perpendicular passes
Machine straight on
Axial offset passes
Up/down milling:
Don't Care
Direction:
Both Ways
Simple ordering
Stock to Leave
Fillets
Smoothing
Feed Optimization

Inventor HSM

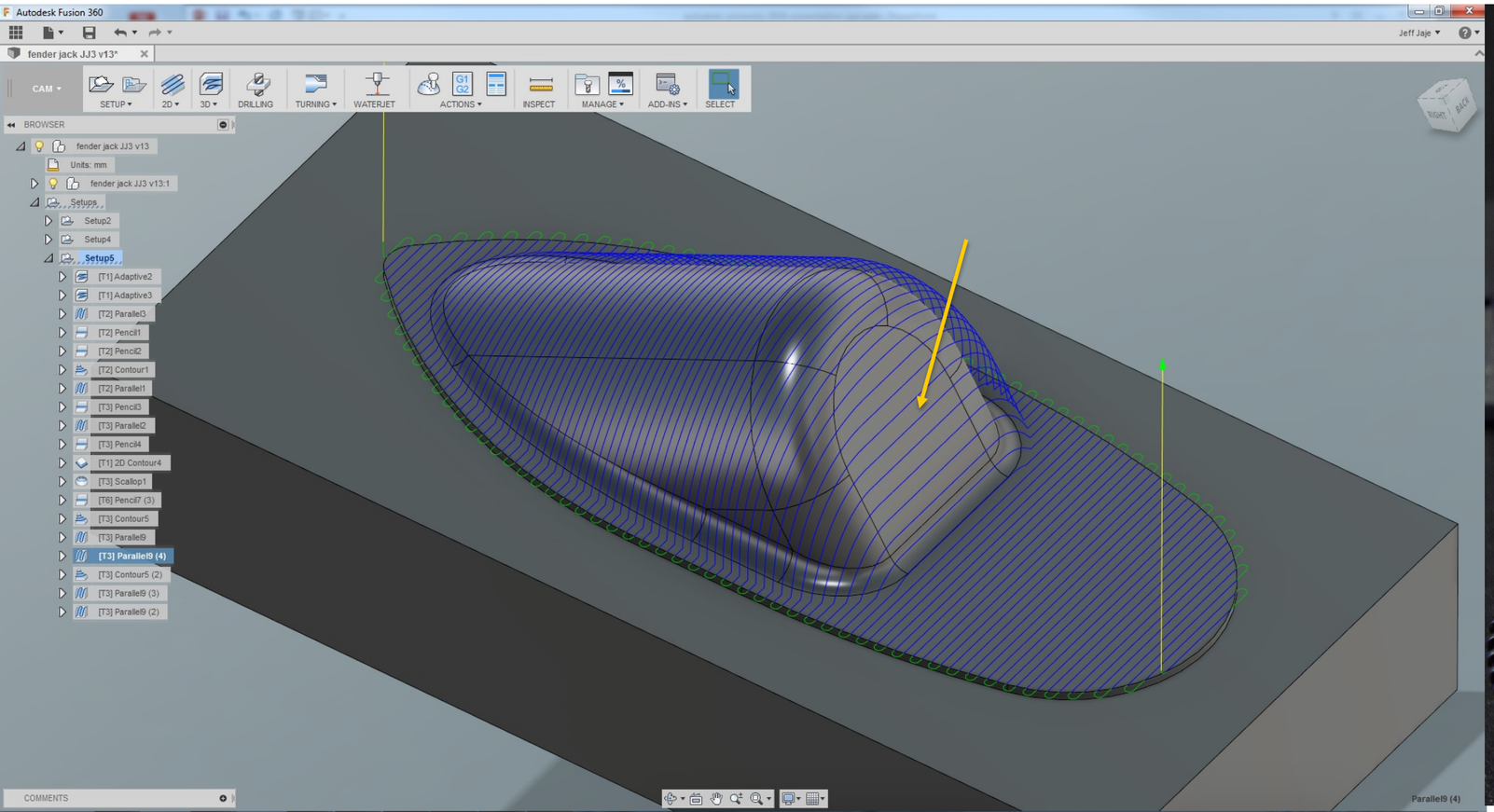


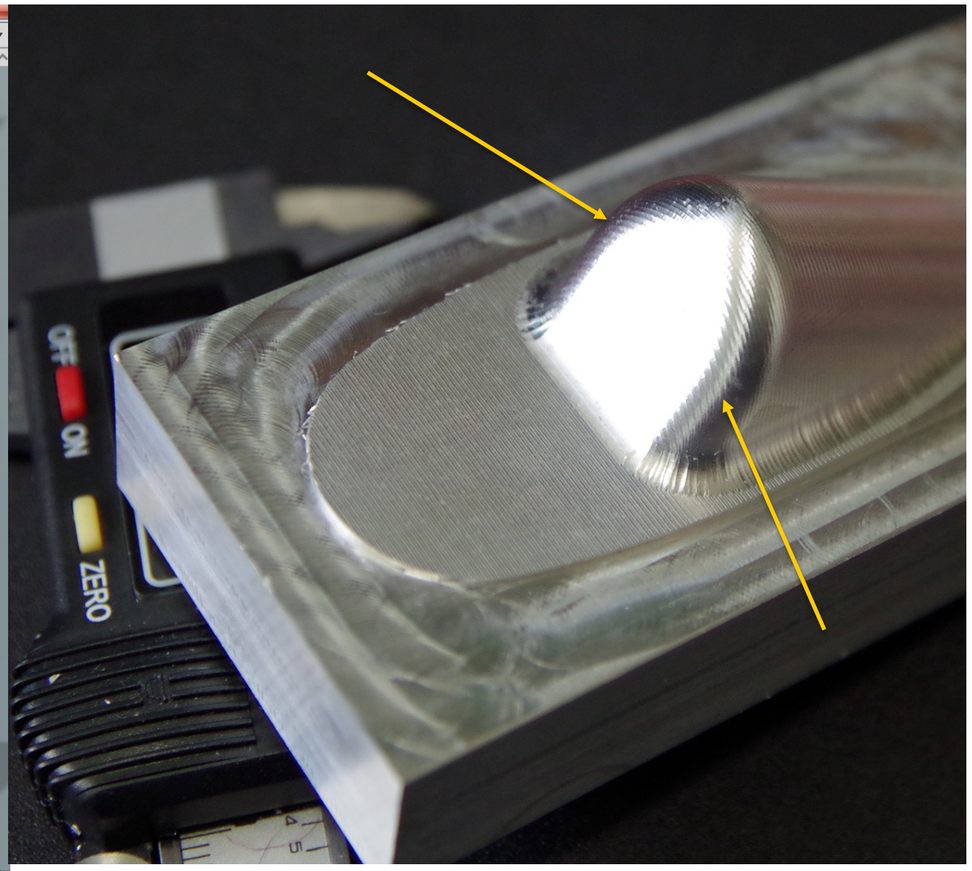
CAM Strategies

Strategy

What strategies you use to program the part are as important as the step and tolerance.

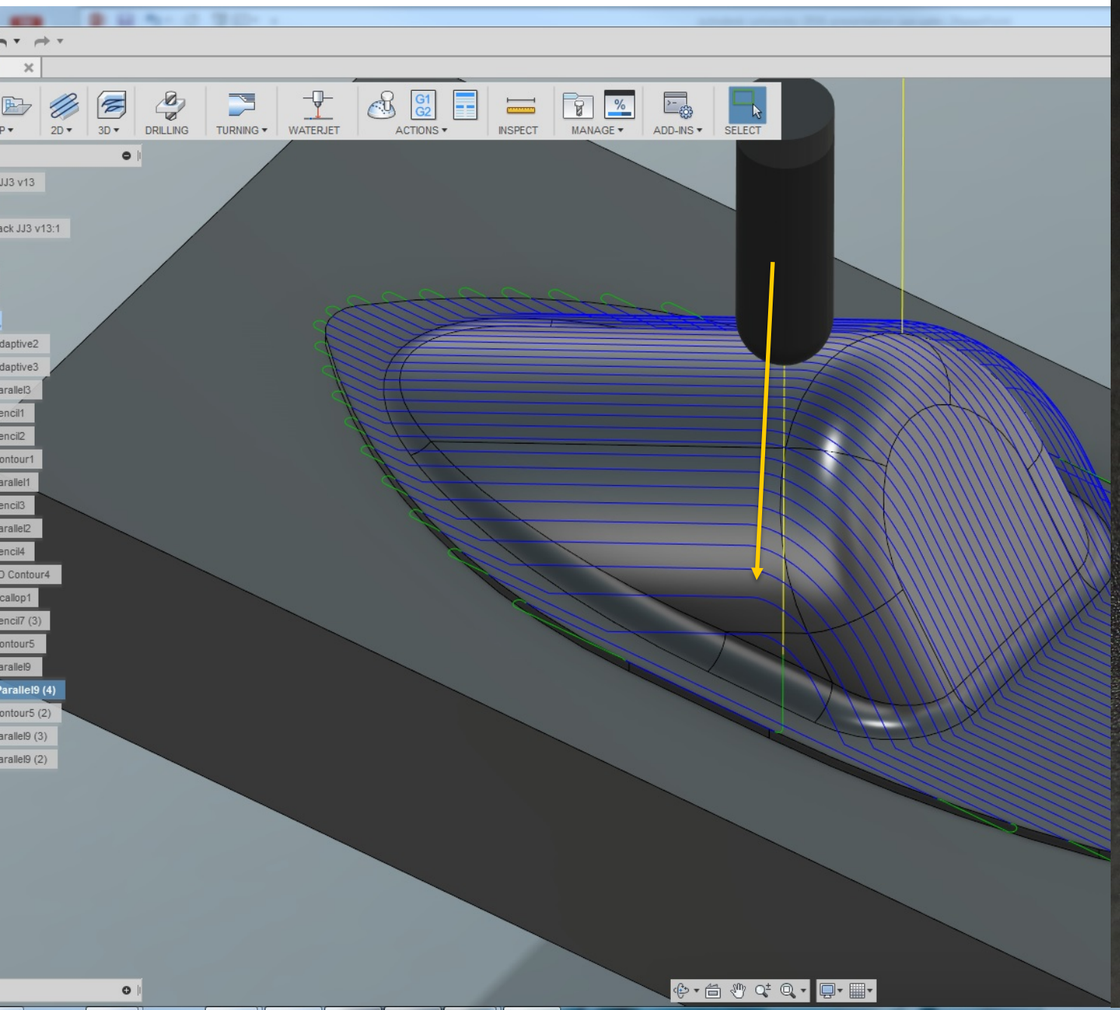
#2 0.25 step



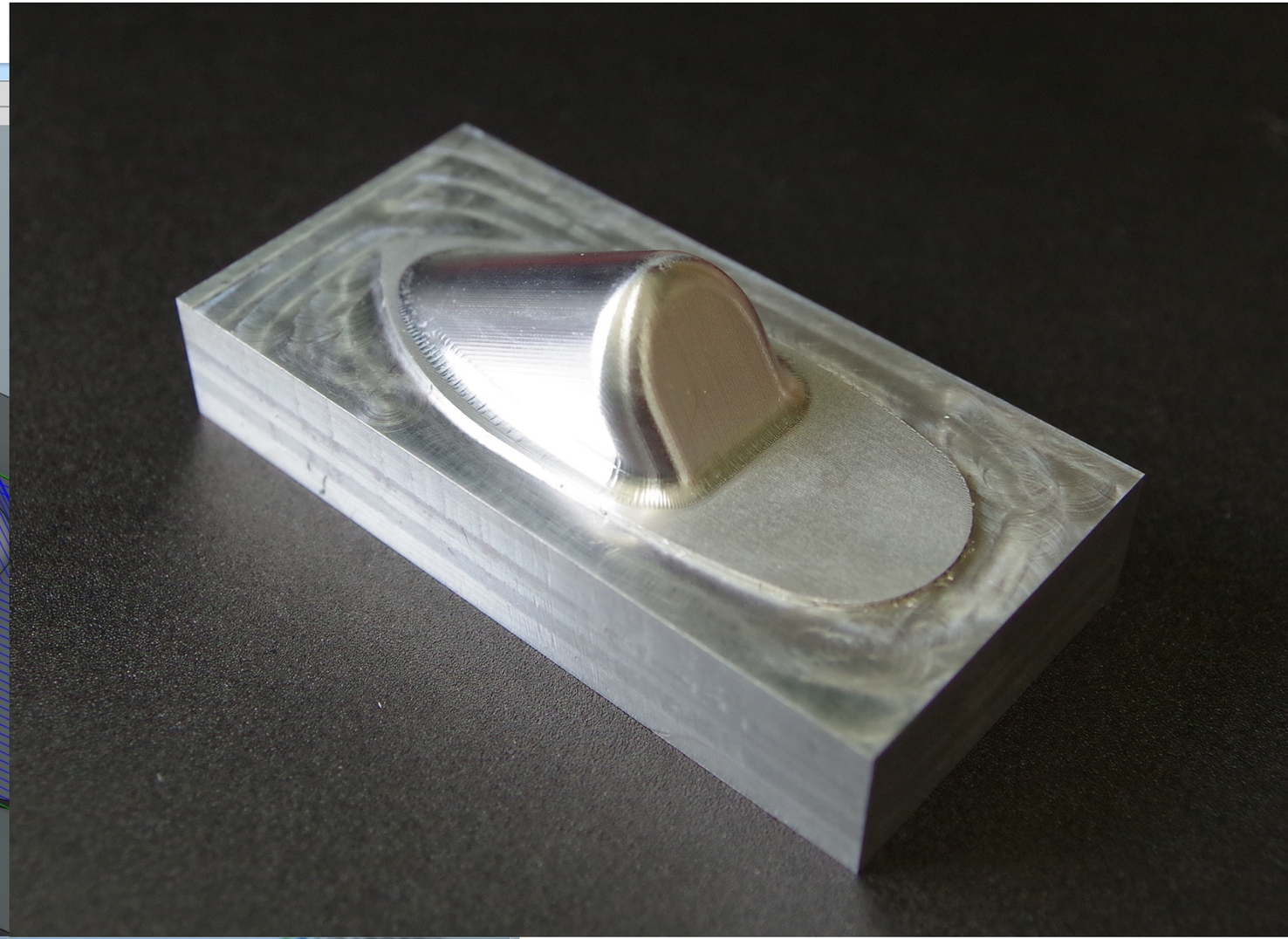


Strategy

Maybe try the other direction.



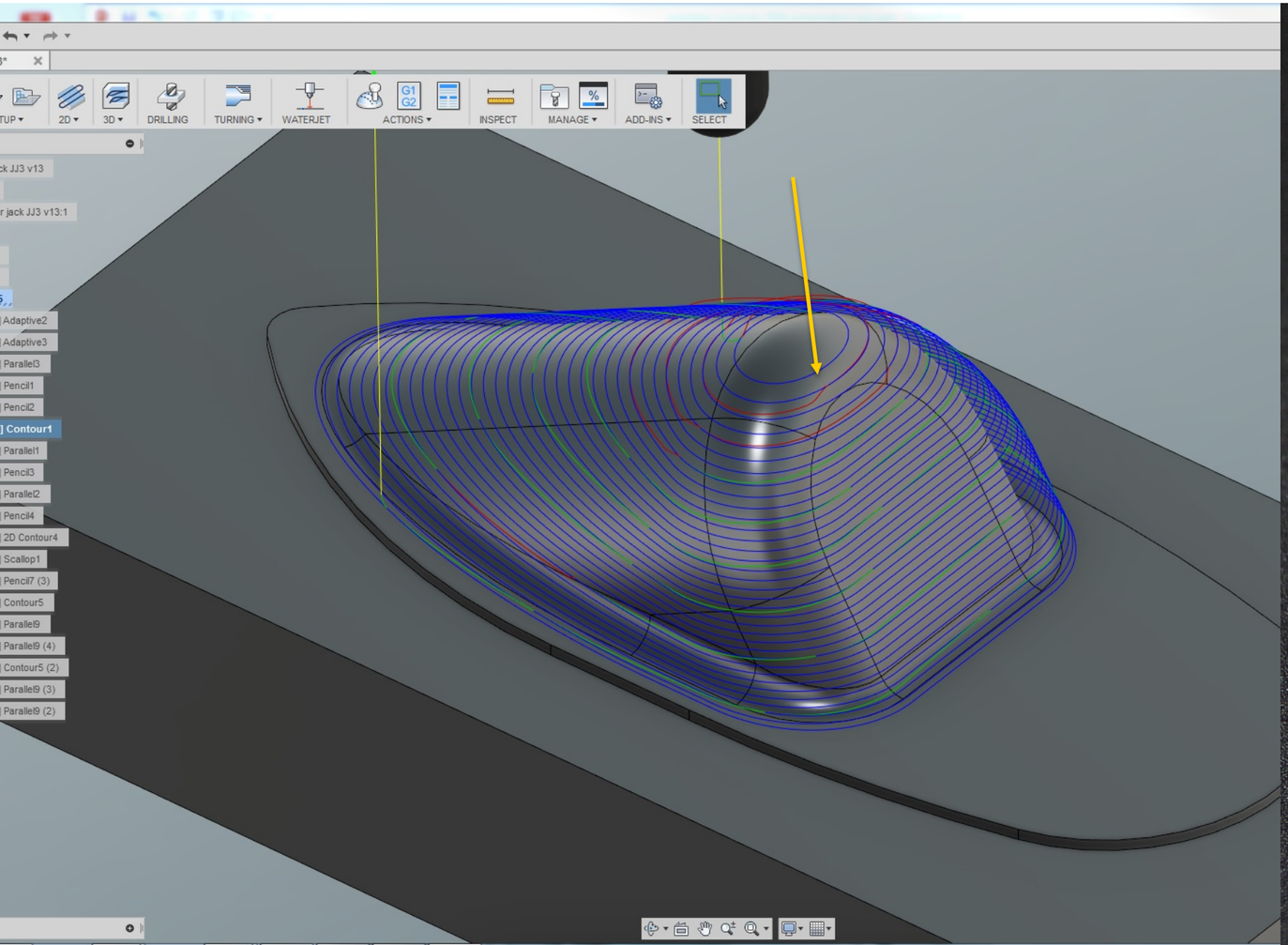
#6 0.25 step 0.0025 scallop

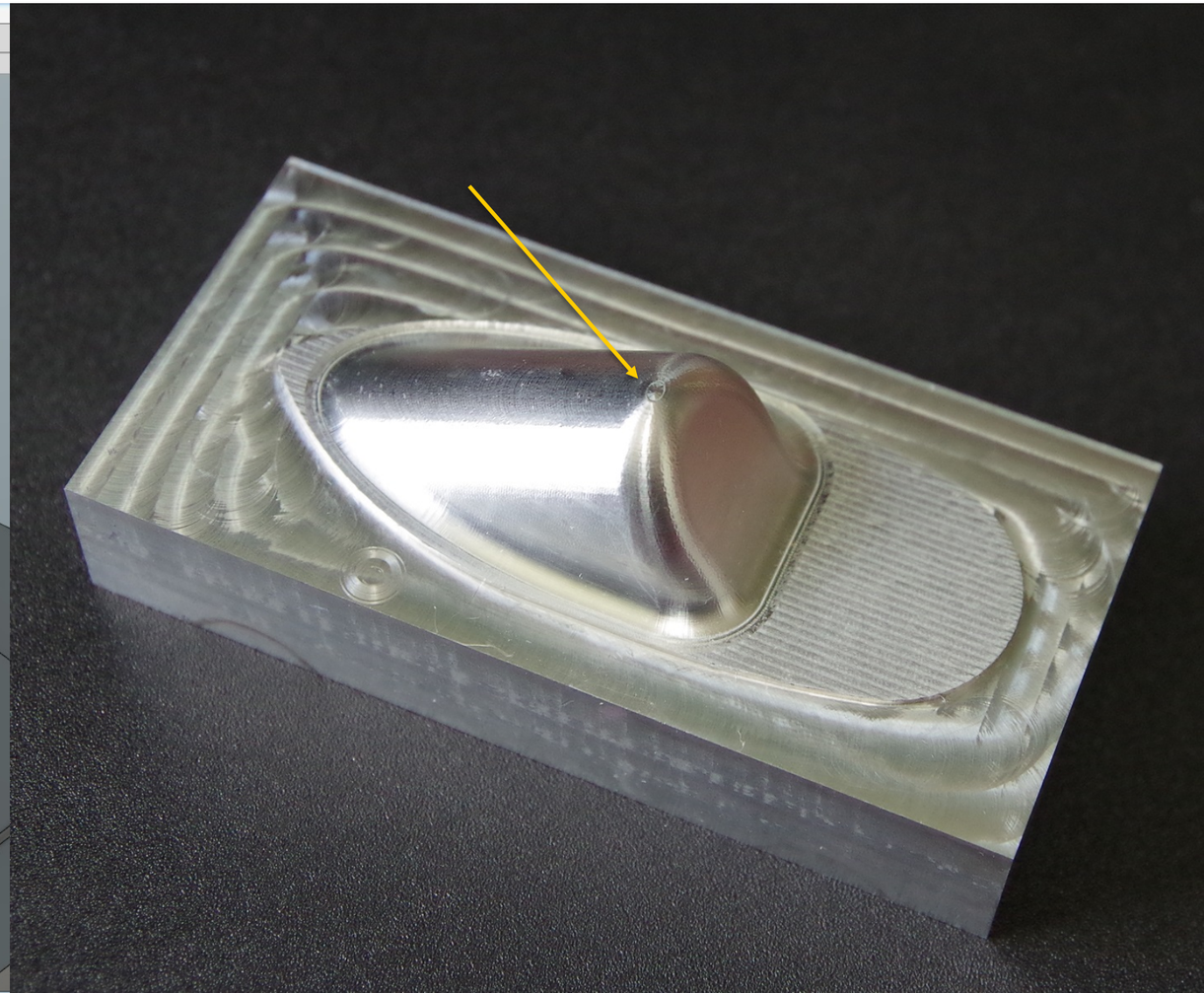


Strategy

Top Down (Contour or Z-Level Finish)

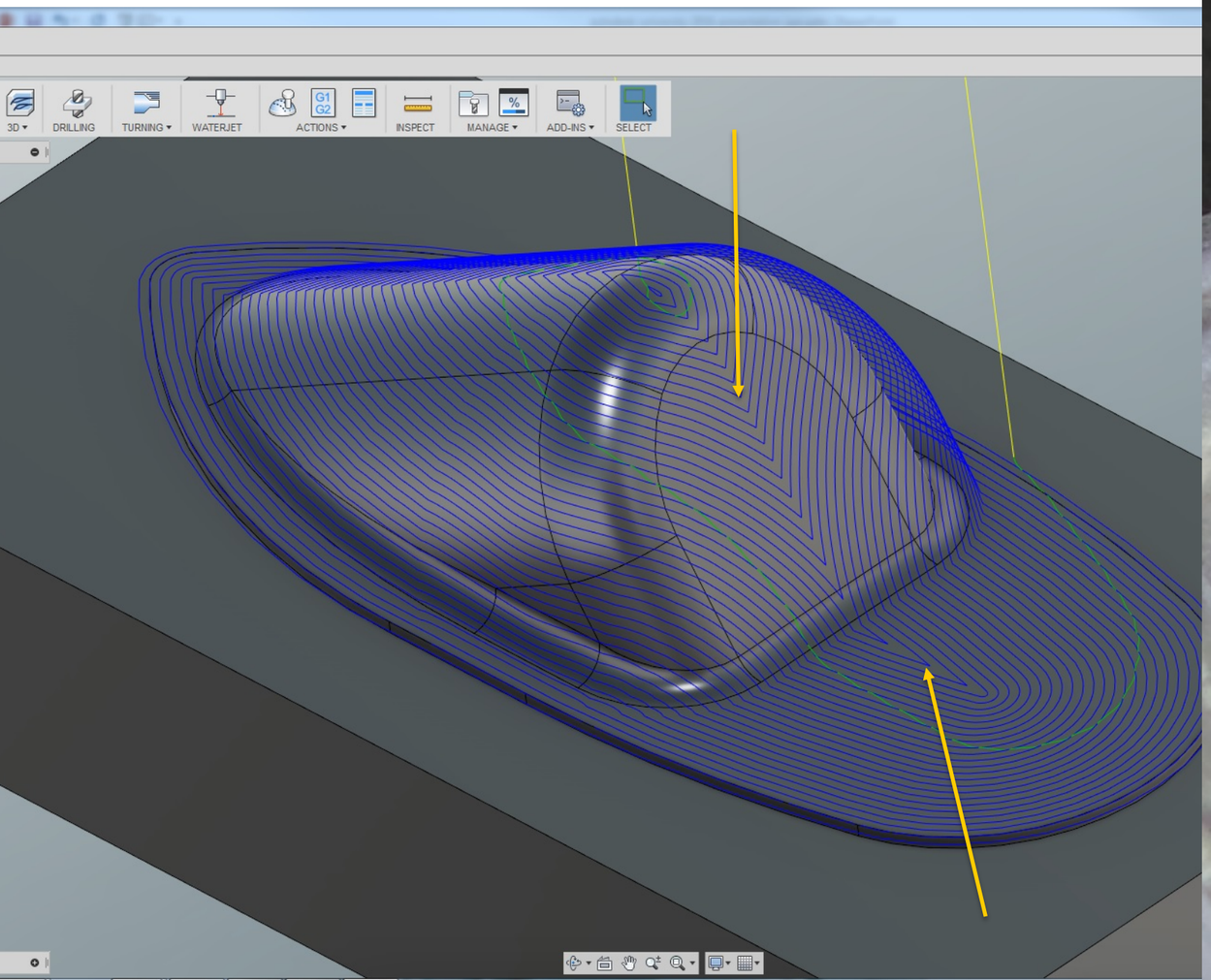
#8 0.15 step down



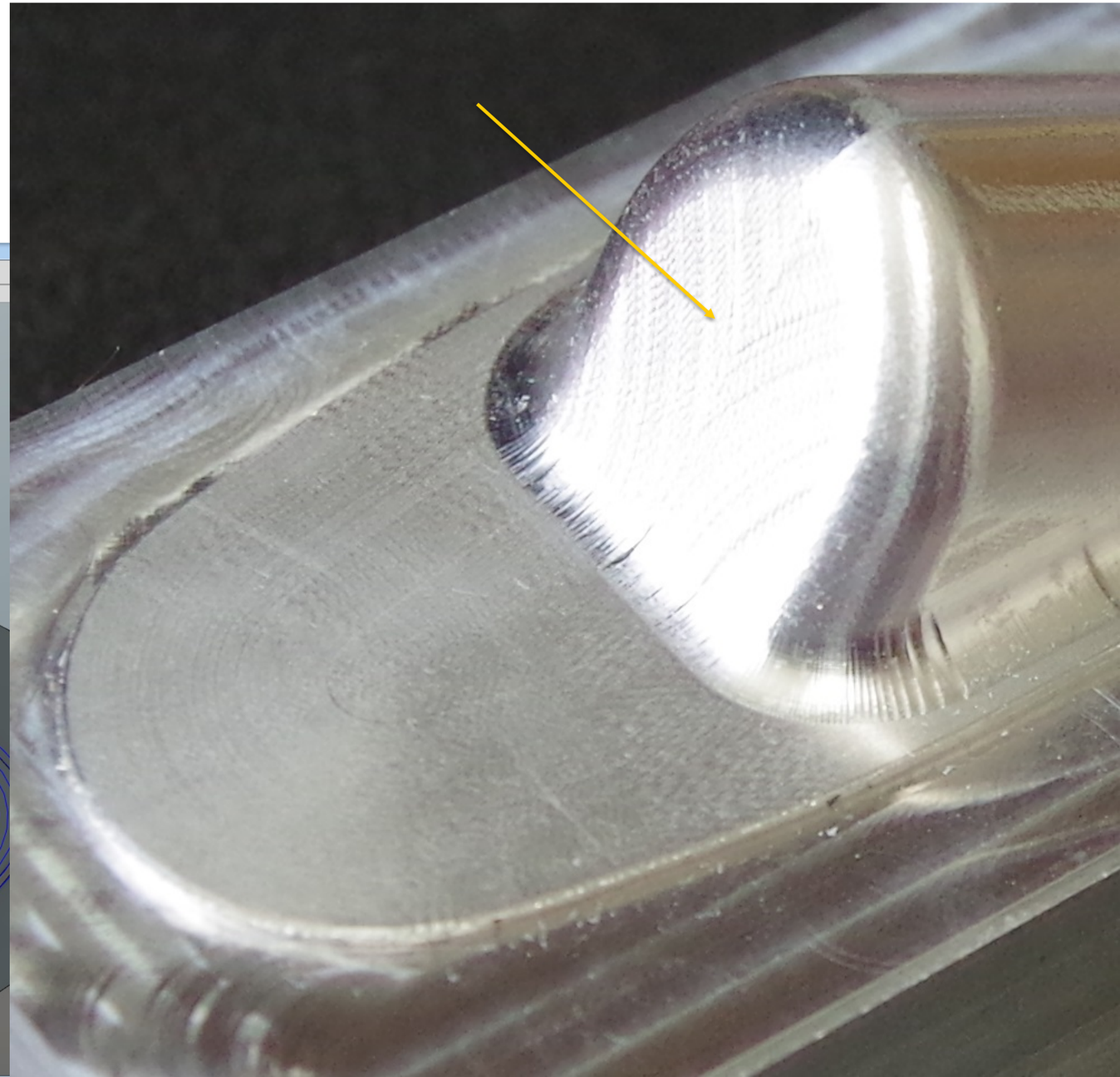


Strategy

Constant Scallop Strategy

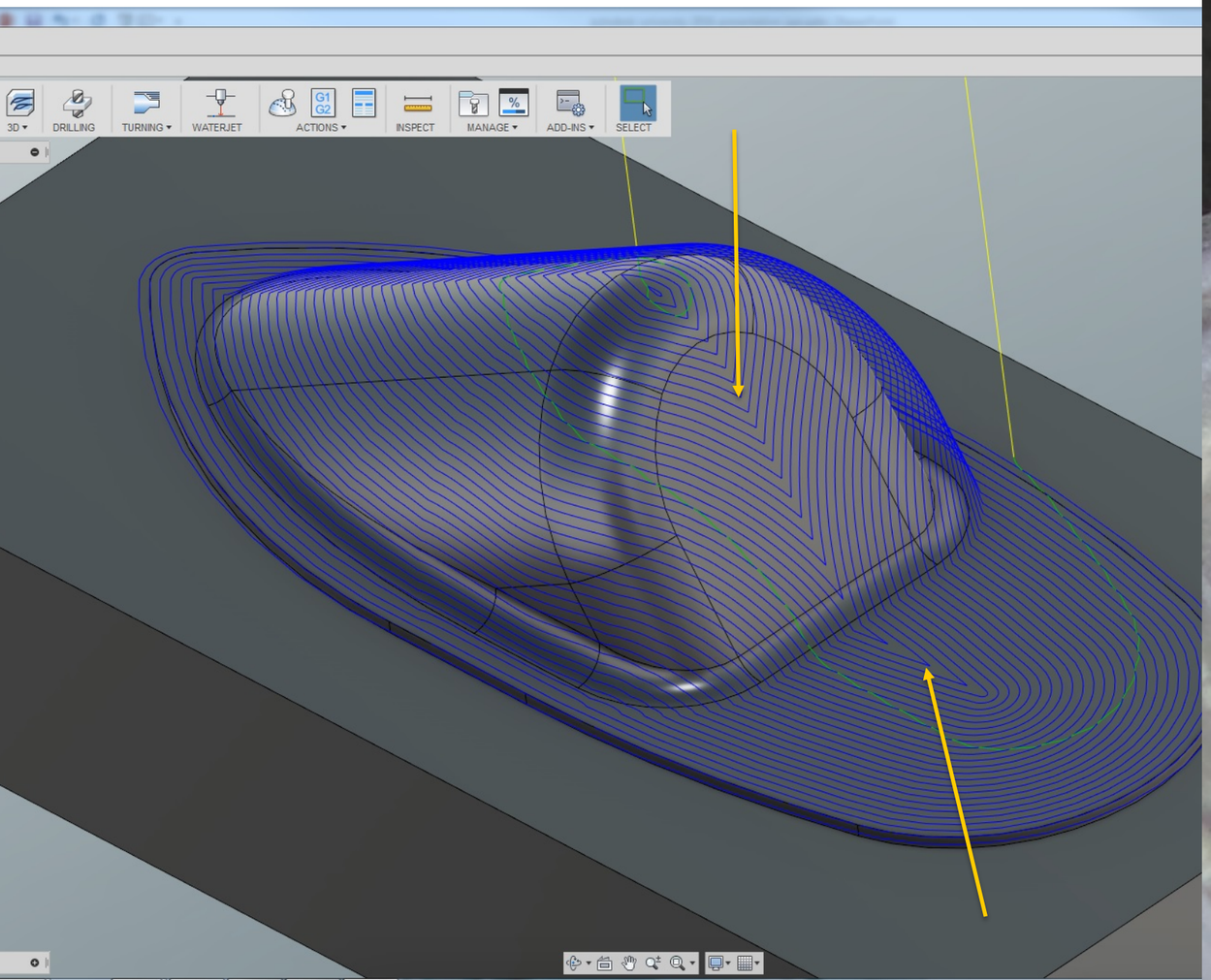


#7 0.16 step “snail tracks”

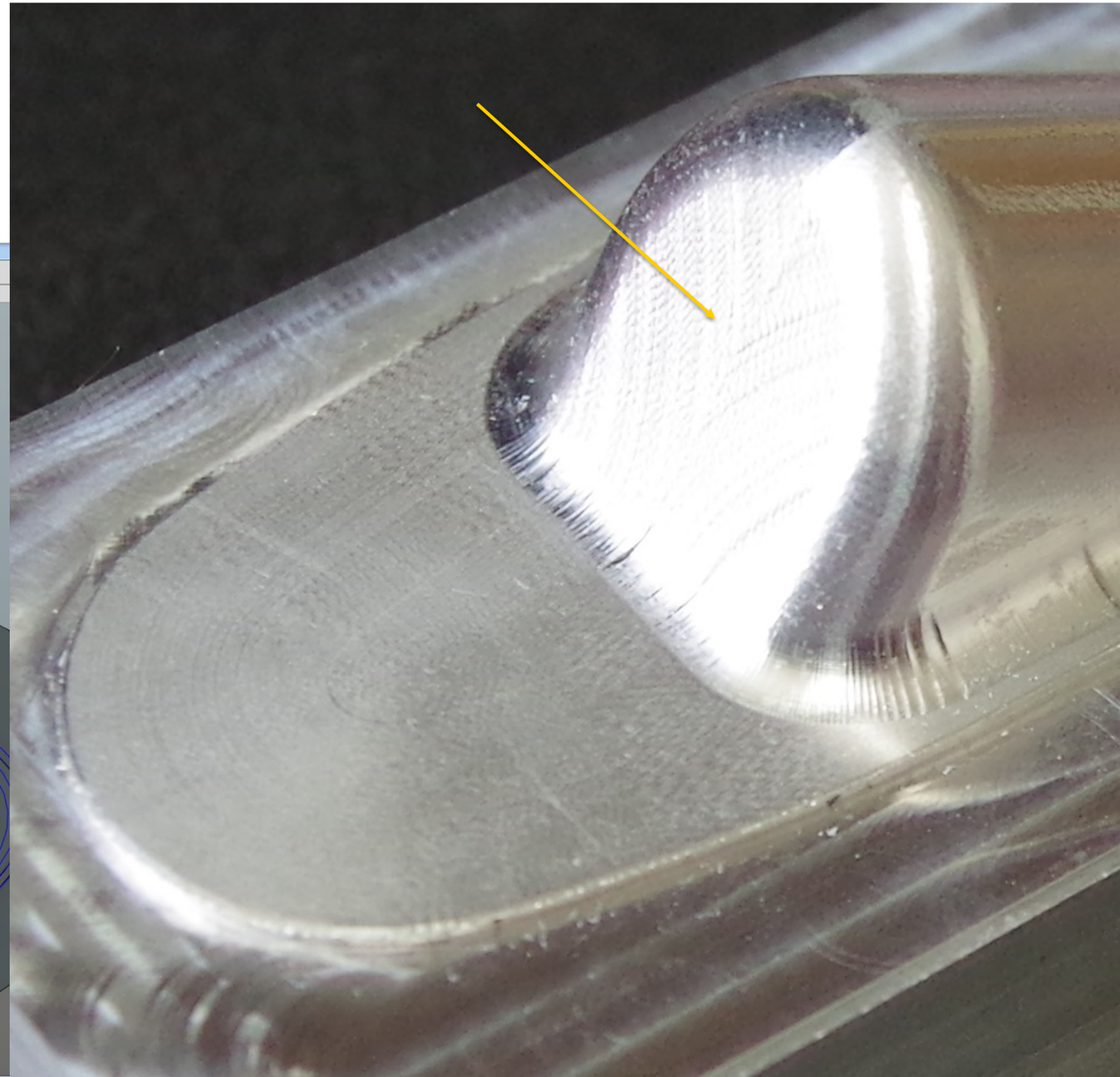


Strategy

Constant Scallop Strategy

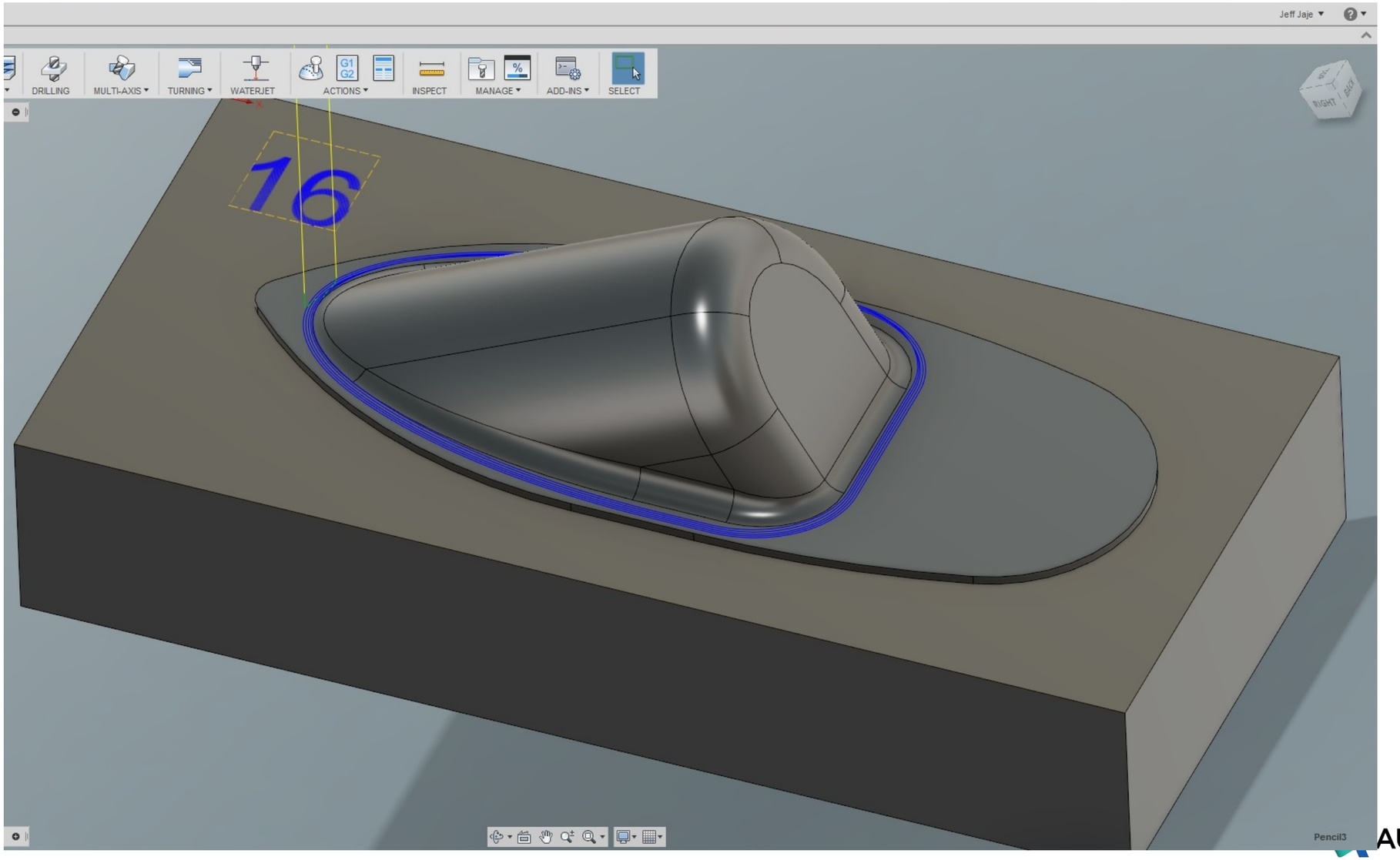


#7 0.16 step “snail tracks”



Strategy

Pencil Trace – before you finish to reduce vibrations in concave corners



Tools

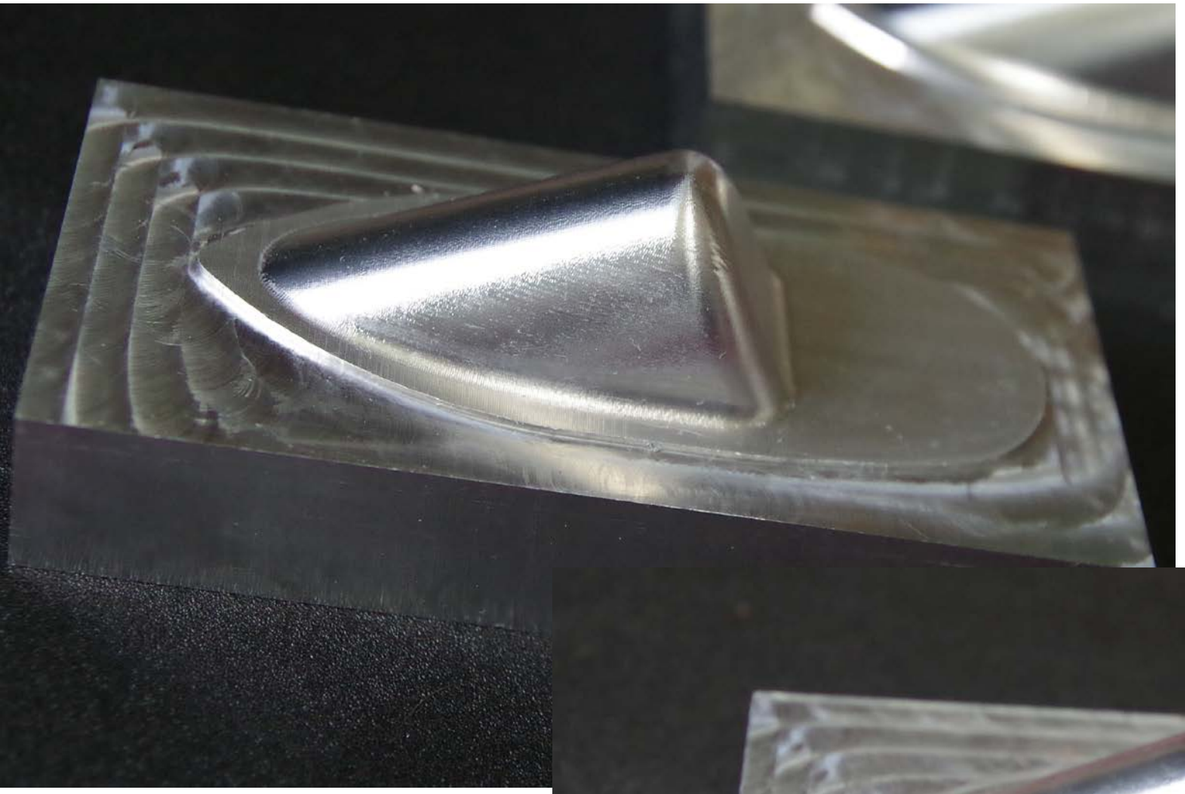
Tools

- The more rigid the tool the less deflection
- Shorter tools deflect and vibrate less
- Strong tool holders allow for stronger tools (shrink fit)

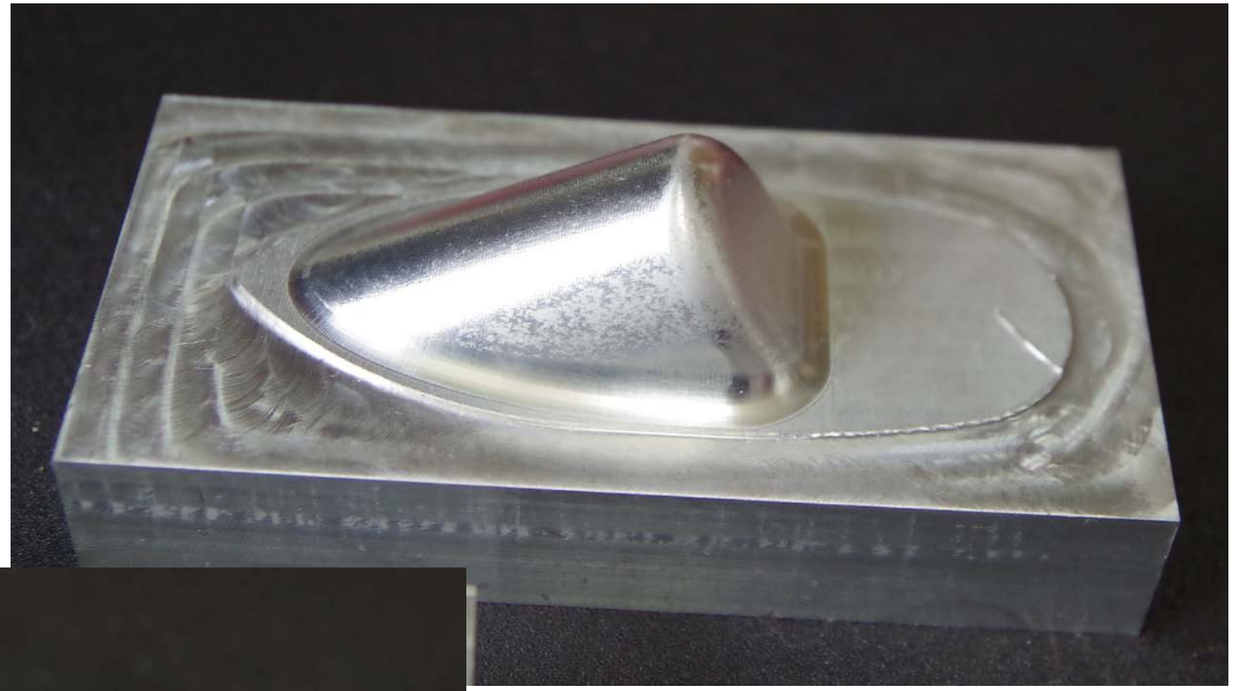
- The bigger the radius, the larger the step for the same scallop (means you can step less, finish faster)

- The more rigid the tool, the faster you can run and avoid effects of deflections and vibrations.

Tools



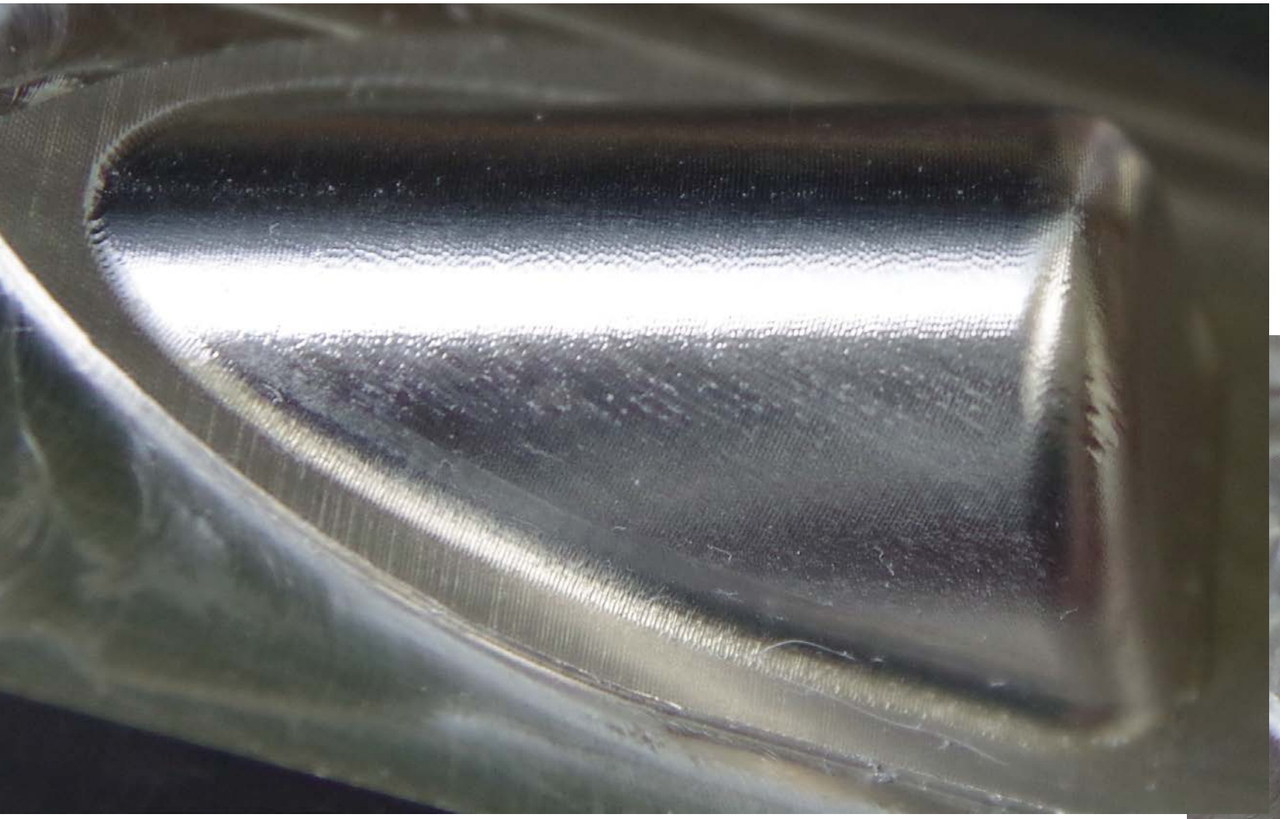
#11 3.175 Ball



#9 9.525 Ball

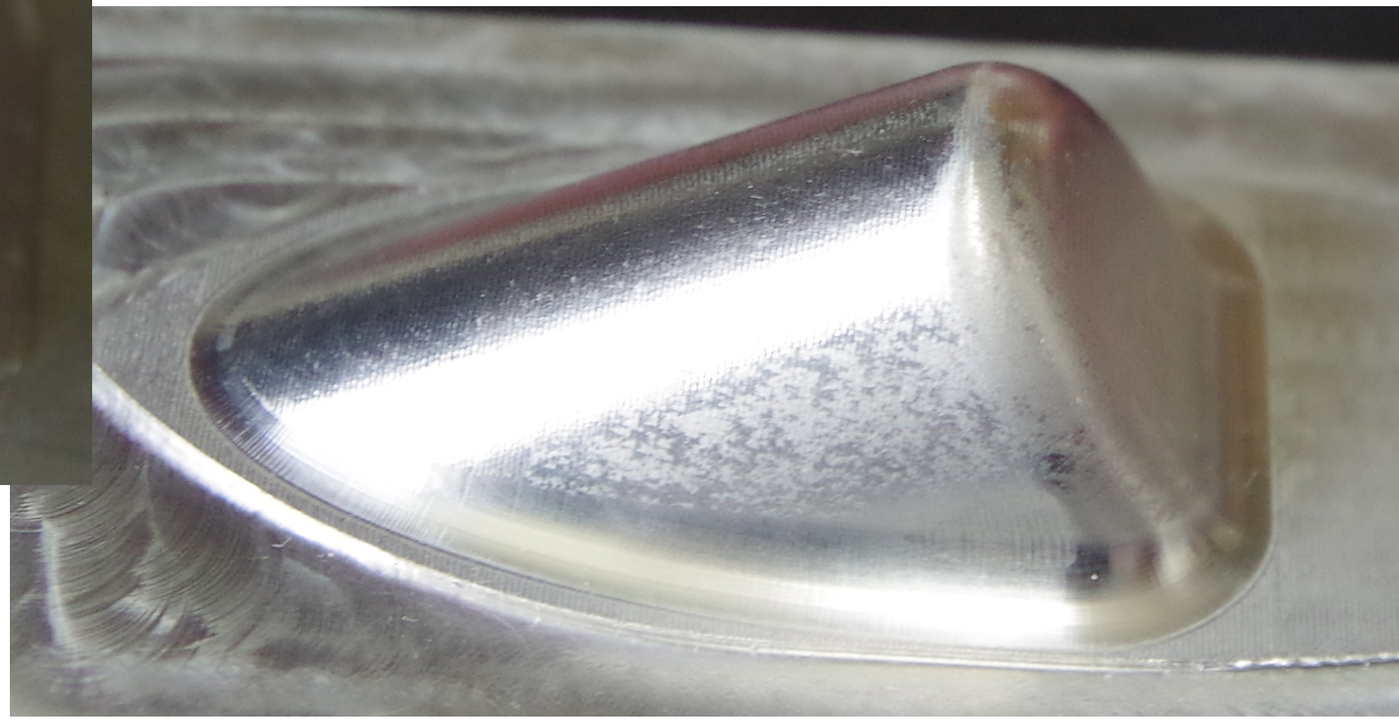
#4 6.35 0.25 Ball

Tools



Orange Peel
You can get interesting harmonics if you use a 5 flute

Could probably run faster



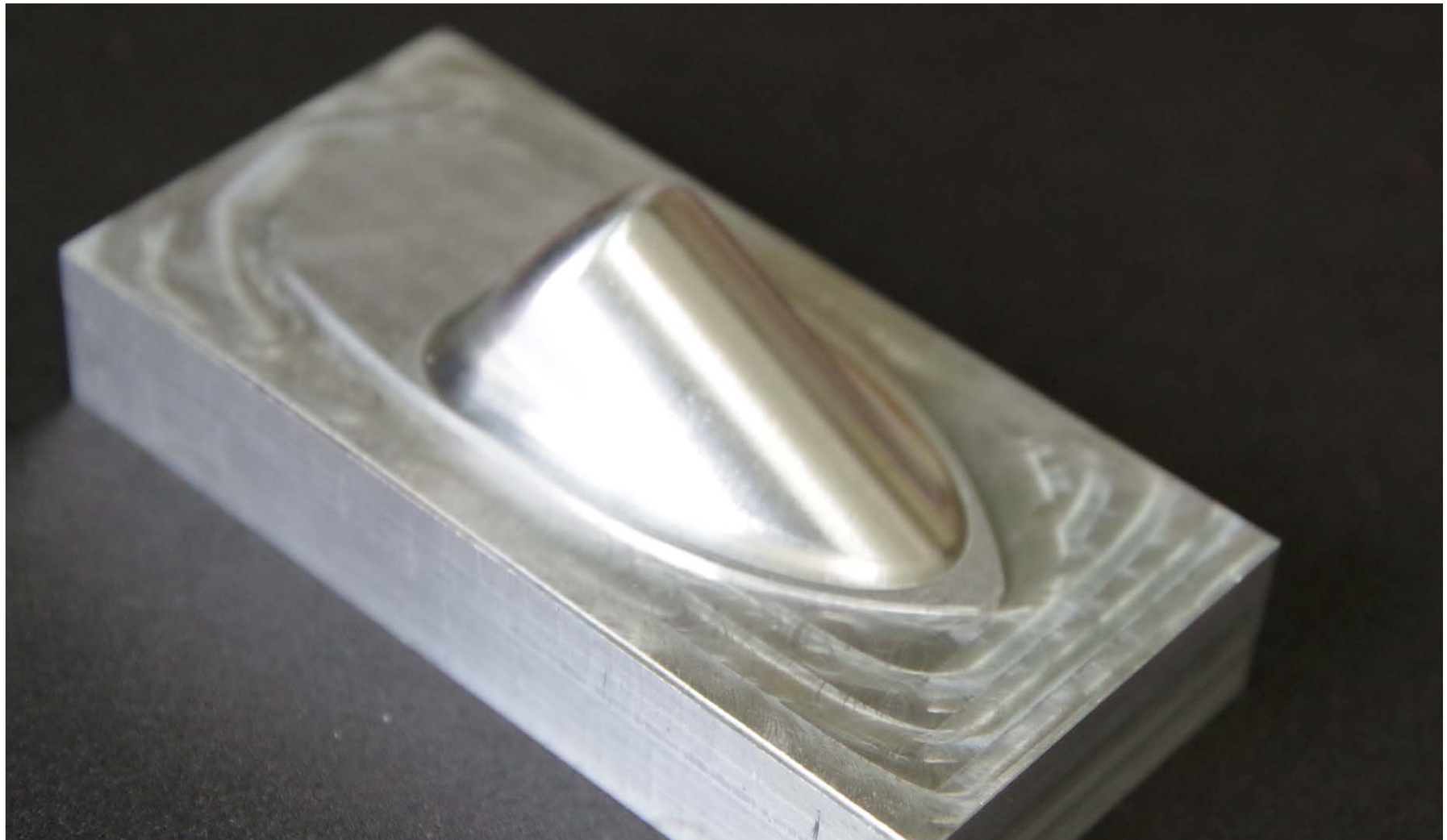
Tools

Many machinists recommend the inserted contouring cutters

Tools

Many machinists recommend the inserted contouring cutters

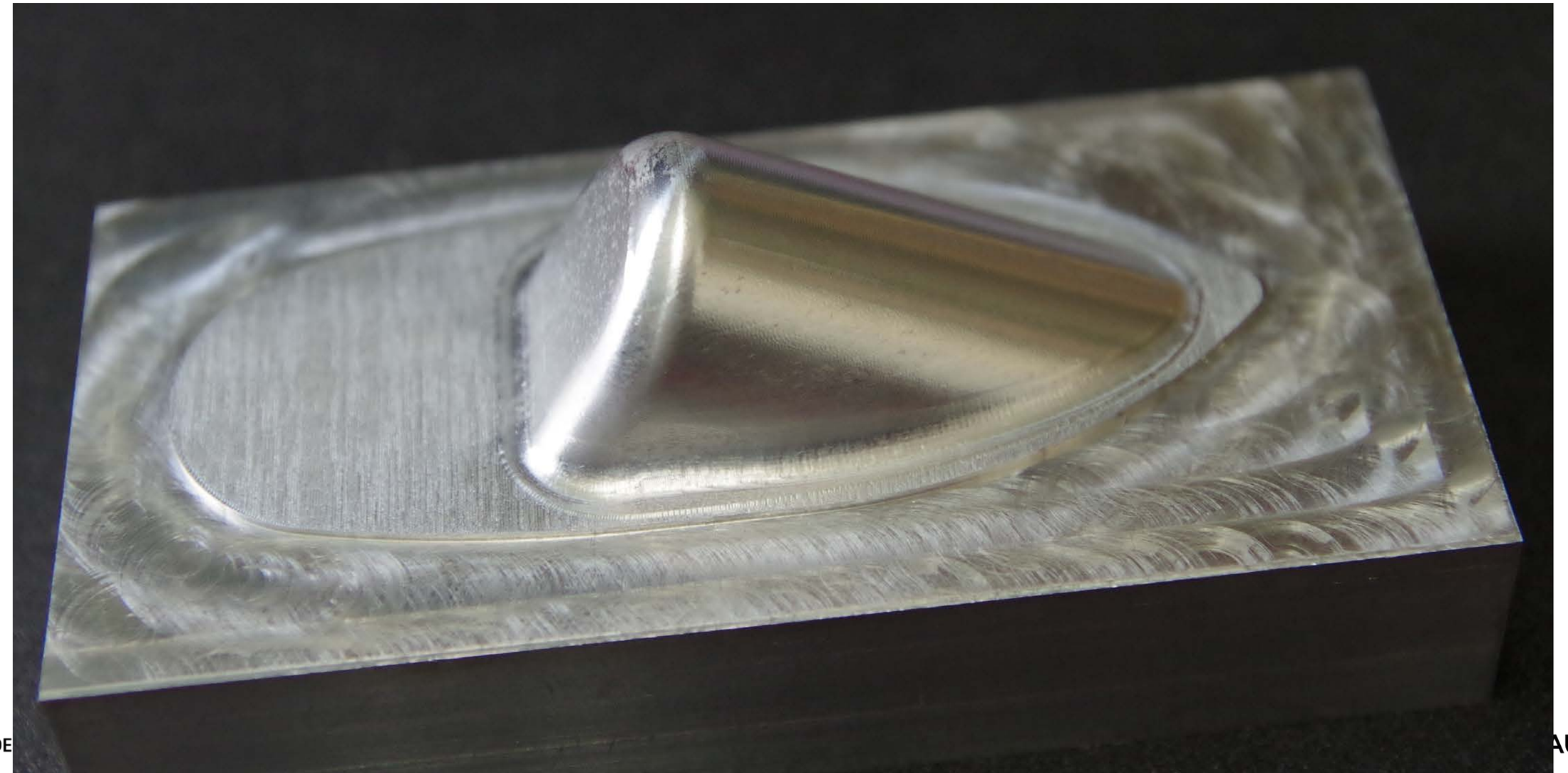
#10 8mm ball



Tools

Tool plus strategy

#12 0.25 step 0.0025 scallop



A Story

How my friend Dave the Machinist got cut 5 strokes off his golf game. Dave is a one man job shop, and for every year, he makes tooling for creating small emblems for cars.

A Story

Dave would load and cut 10 molds at a time, and play a round of golf. He's return. Squirt the pieces with polishing compound and a 1/8" polisher, let it go, and play another round of golf.

A Story

And that’s how Dave dropped 5 strokes from his golf game.

#15 polished

Questions?

Special Thanks to:

More Questions? Visit the AU Answer Bar & Manufacturing Area of the Exhibition Hall

- We have mills there.
- We have our expert Application Engineers
- Titan Gilroy is there and would love to talk shop

How did I do?

- Your class feedback is critical. Fill out a **class survey** now.
- Use the AU mobile app or fill out a class survey online.
- Give feedback after each session.
- AU speakers will get feedback in real-time.
- **Your feedback results in better classes and a better AU experience.**