

August 26, 2020

White Paper

Understanding CAD/CAM-Generated Meshes

CAD/CAM packages share a common feature that needs to be looked at and fully understood. All packages create what we call a tessellated mesh file for machining. You do not machine the solid model; you do machine the tessellated mesh file.

What is a Tessellated Mesh File?

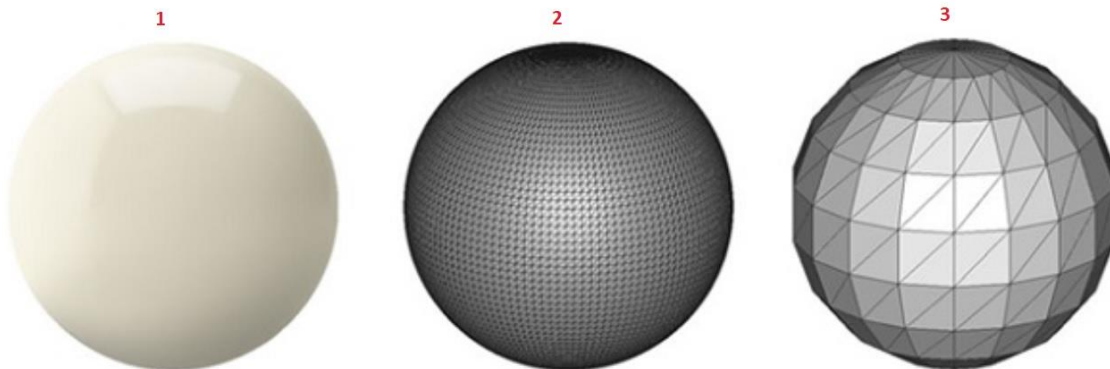
Tessellated Mesh Files contain information derived from 3D models. This format of adjoining triangles describes only the surface geometry of a three-dimensional object without any representation of color, texture or other common model attributes. Tessellation is the process of tiling a surface with one or more geometric shapes such that there are no overlaps or gaps. A tiled floor or wall, that is a good real-life example of tessellation. STL files used in 3D printing are tessellated mesh files.

What You Need to Know

In your CAD/CAM system, the tessellated mesh file is completely transparent to the end-user. The file's accuracy and usability are a direct result of two critically important inputs from you:

1. Tolerance – roughing and finishing
2. Arc Fitting or Smoothing – roughing and finishing

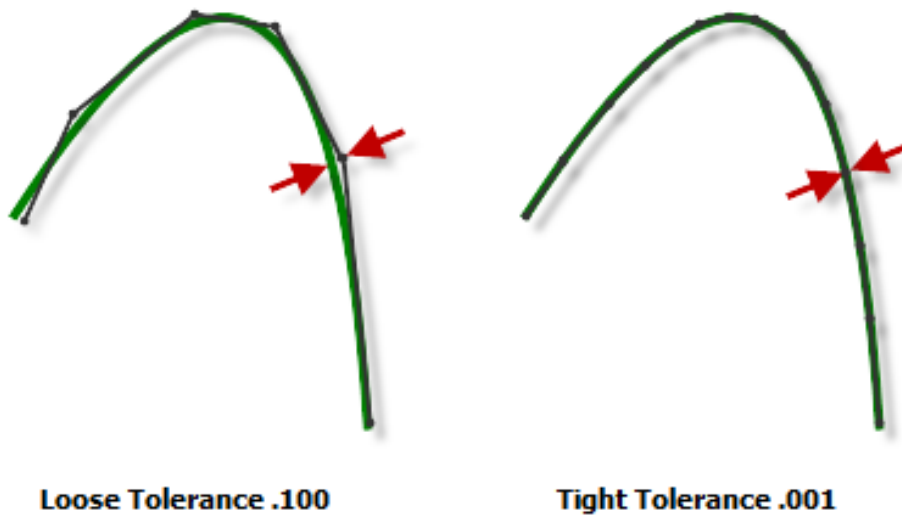
Let us first examine the effects of Tolerance on a model.



The perfect solid spherical model (1) on the left is approximated by triangular tessellations. The figure on the right (3) uses a loose creation tolerance, resulting in a coarse, faceted model. The figure in the center (2) uses a tighter creation tolerance giving smaller triangles and achieves a smoother, more accurate workpiece approximation (source: i.materialize)

The Tolerance parameter controls the accuracy of the mesh model that is created to drive toolpath generation. The b-rep geometry in the file is “meshed” and sent to the CAM kernel.

The “mesh” of the part created during toolpath generation defines the linear segments of the toolpath. The allowable variance between the true solid model geometry and the mesh as a chord distance as defined by the toolpath Tolerance parameter.



(source: Fusion 360)

A common misuse of tolerance often occurs during the roughing process. Having too tight of a roughing tolerance can result in too many moves to describe things as simple as a straight line. Instead of having a start and end point that represent a line, there could be hundreds of small linear segments making up that simple line. This scenario drives up the file size and drastically increases the data processing (toolpath creation) time. A loose tolerance will generate quickly and give smaller file output sizes but may not follow your solid model geometry as accurately. This is often tolerable in part roughing as there is still finish stock to consider for final clean-up.

A tighter tolerance will result in a more accurate toolpath but also in a larger NC code output file due to the increased number of linear segments – each segment is a line of code. Toolpath generation and data processing times will take substantially longer. Tighter tolerance is, for the most part, most desirable for finish machining. For finish machining, you will need the toolpath to more closely represent the model.

Can I Reduce My File Size?

Although a files size is not an issue with your ProtoTRAK control, its size does influence data processing time. As a solution, many CAM packages allow you to convert data in the G17, G18 and G19 planes into arcs, within a user-given tolerance. So, rather than having 40 – 60 G1 linear segments representing a section of an arc, you could have but one (1) single G2 or G3 arc move. That alone will drastically drive down the file size & data processing time. Some CAM systems call this arc-fitting while others call it smoothing. Let’s examine a smoothing picture from Fusion 360:

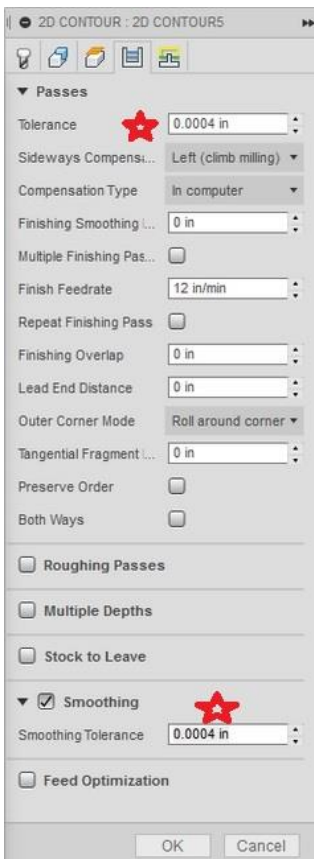
While Tolerance controls the accuracy of linear segments of the toolpath, the Smoothing parameter fits arc moves to your toolpath within your specified tolerance.



Smoothing Off

Smoothing On

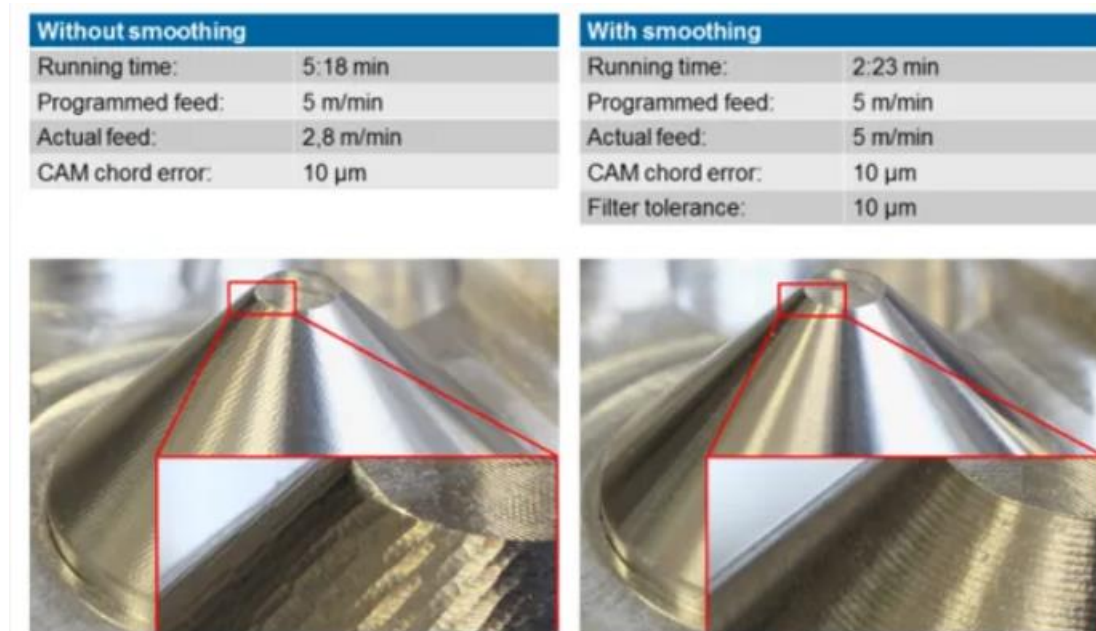
Notice the 'Smoothing Off' illustration has 15 nodes, while the 'Smoothing On' illustration (arc-fitting) has but 5 nodes. The result is 2/3rds less data to process.



If you examine this Fusion 360 parameter sheet, you will notice (at the red stars) that the Tolerance and the Smoothing Tolerance are identical. While these values may be fine for finishing, they are a poor choice for roughing.

Try loosening your Roughing tolerances and watch your file size and processing times decrease. Remember, you have finish stock to take up for the looseness of the roughing passes. Your roughing tolerances should be gaged off your finish stock amount, so allow enough stock to clean up the part to its desired shape.

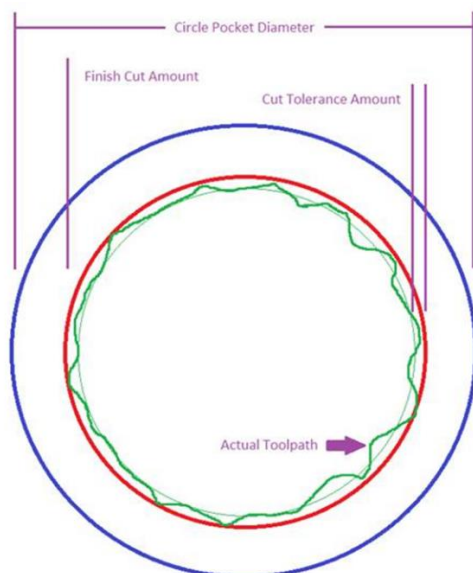
It will take some practice and experimentation on your part, but you will find that it will become time well spent.



(picture source: Datron)

Shown above is a common error made by many programmers. Notice the roughness of the finishing pass in the picture on the left. It does not have any Smoothing callout, and the CAM chord error (Tolerance) is set to 10 μ m, or about 0.00039". That tight of a tolerance creates a large tessellation file. To decrease the CNC file size, as well as calculation & run time, there should be a Smoothing (Filter Tolerance) value defined. The picture on the right side is much better, as it utilizes the same tight Tolerance (CAM chord error) as the one the left, plus a Smoothing (Filter tolerance) tolerance of an equal amount. The actual deviation from the nominal model shape is a combination of the Tolerance plus the Smoothing values (0.00039" * 2). Your Tolerance and Smoothing values, when used, should always match. The difference in run time can be attributed to the part on the right cutting arcs vs the part on the left cutting a ton of linear moves. Arcs cut faster and smoother than lines.

A Closer Look at the Roughing Tolerance

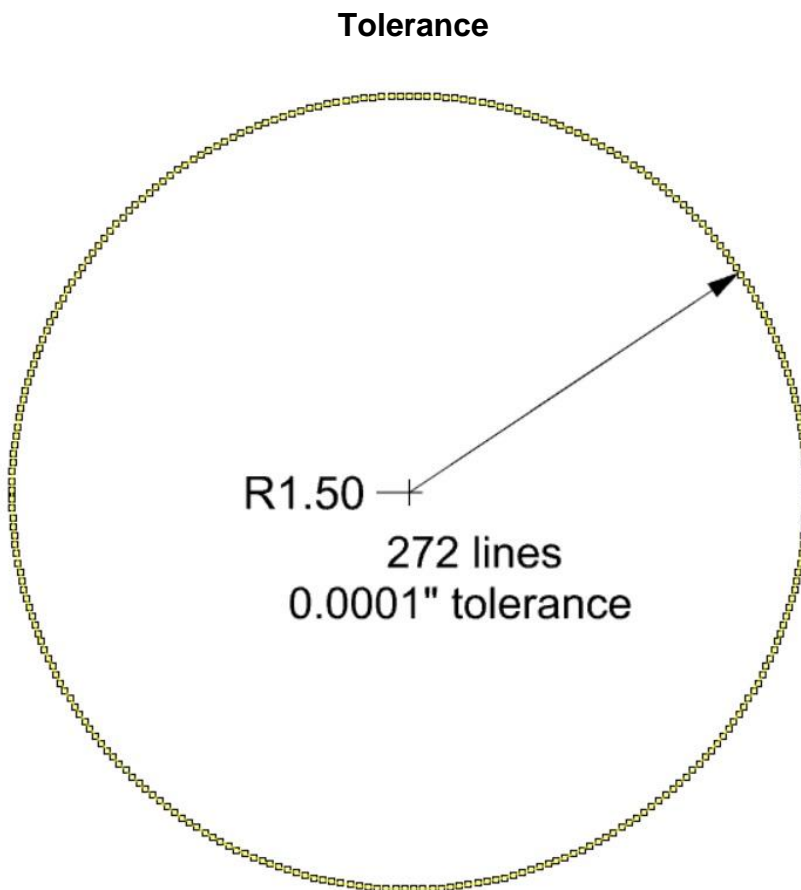


The roughing tolerance can be much larger or looser than the finishing tolerance. The roughing process contains a finish stock amount to protect the finish geometry from rough cutter 'wandering'. You can see the rough cutter path in green and it is held within the finish cut stock boundary in red. A looser roughing tolerance means smaller file sizes, faster processing times and faster data execution while machining.

A Closer Look at Smoothing (arc-fitting)

The following picture examines the effects of smoothing. Both files created from the samples below will run in a CNC machine. However, one file will calculate and run faster with a better surface finish than the other.

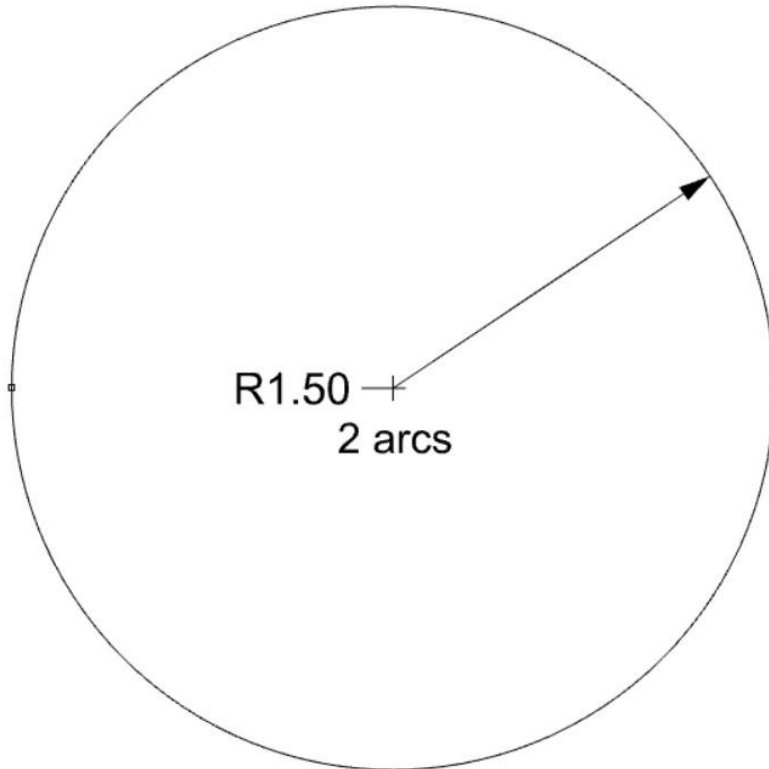
What is sad is that many programmers will create their files so they look like the first picture and then wonder why the program takes so long to calculate, and then the machine 'chokes' (shucks) during machining. And then they wonder why they cannot pull the surface finish they desire. I often hear programmers say: "Hey, I set the machining tolerance to 0.0001". It should have an excellent finish". Those programmers need to understand that there is more to a successful process than simply using a tight tolerance. Sure, a tight tolerance, especially for finishing, is part of the solution, but it is not everything.



A Tight Tolerance

Here you see raw data as generated by a CAM system. This simple circle can contain up to 272 linear segments (connecting the points) that were created when a 0.0001" fitting tolerance was used. Each linear segment must be calculated, processed and then executed during machining. **This IS NOT efficient.**

Smoothing



Arc-Fitting

Here we see the results of arc-fitting. The file now contains only two (2) 180-degree arcs. Consequently, extraordinarily little processing time, faster execution of the data and a much higher quality surface finish are the results. **This IS VERY efficient.**

Conclusion

Learning through experimenting with your CAD/CAM system's tolerances and smoothing parameters can make the difference between creating either a desirable toolpath or a near-useless toolpath. Making poor or uneducated choices in this one area alone can increase machining run time, polishing time, toolpath calculation time and can even make for inaccurate parts.

Spend some time learning how your CAD/CAM system is setup in these two areas and learn how to best utilize these two items so they become desirable features and not roadblocks to making well-manufactured parts.

For roughing, start by opening your tolerances and for finishing, start by cranking down those tolerances. Do some experiments on curvy parts to see what is needed to eliminate faceting and maximize surface finish. Also, try to find the limits of your ProtoTRAK controller. Look for the point where you are feeding it so much detail that surface finish starts to degrade. These are variable but valuable pieces of information, and they can only be learned by trial and error.

(partial source: CNC Cookbook)