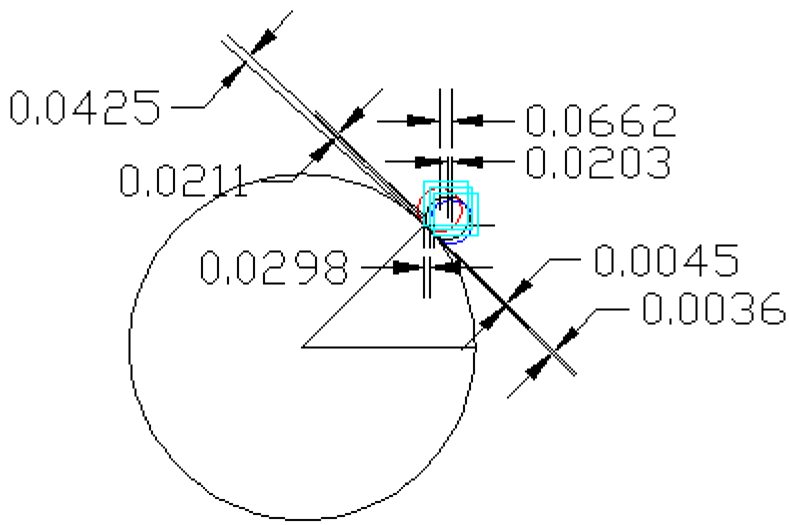
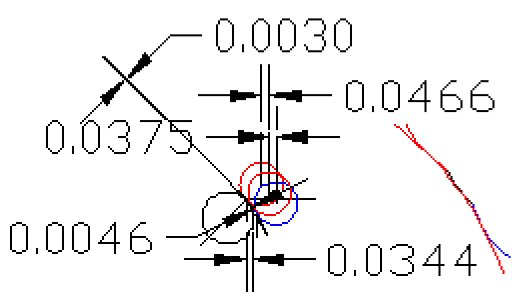
PIC OF CHECKING THE STEPOVER FOR THE MINITURE BARREL FOR THE SMITH

I actually drew it out in ACAD. My choice was sacrificing machine time for finishing time. Thus I got down to about 0.002” which is almost not noticeable and only required a few light strokes of the sandpaper. Just draw the tool tip out at different stepover’s and see how it tracks the circular part of the piece.





How to Choose a Stepover

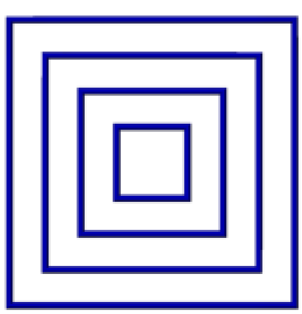
Posted by Robert on May 16, 2011 in How To | 0 comments

One of the fundamental parameters of any CNC machining, and 3D machining in particular, is the **stepover**. It is not a stretch to say that it is the single most important parameter in determining the quality of the finished parts you will produce. A machinist can pick a value by feel, based on previous experience, or do the math and calculate the exact value that will give them the finish required. New users generally don't have the experience and don't know the math so it takes a while to get an intuitive understanding of of the stepover parameter.

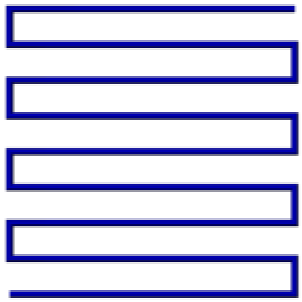
The following post focuses mostly on 3D toolpaths so we'll be assuming the use of a ball mill. Once you understand the basic concepts it's easy to apply them to flat end mills and bull mills. We'll try to build to some rules of thumb rather than derive equations that most users won't be interested in.

Definition of Stepover

Almost all CNC toolpaths are based on the concept of one toolpath being offset from another by some distance; this offset distance is generally called the stepover. Most CAM software, MeshCAM included, uses a couple toolpath styles in particular with these offsets- the raster toolpath (sometimes called a zig-zag toolpath) and a contour offset.



A contour offset toolpath



A raster or zig-zag toolpath

Adjacent sections of the toolpaths above are separated by the stepover value chosen by the user.

Scalloping

The pictures above show how a toolpath is arranged from above but a side view clearly shows the primary side effect of your stepover choice- scalloping.

Scallop shown in red between adjacent passes

The area in red is the part of the stock leftover on the part in between the toolpath offsets. It's important to understand that these are not good; they are not in the CAD and may need to be removed after machining by sanding or polishing. CNC machinists are almost always trying to reduce the scalloping as much as possible and many man-years of effort have been spent trying to develop toolpath algorithms that minimize them.

Scallop vs. Stepover

A moment spent looking at the image above illustrates at connection between scallop height and the stepover value- increase one and the other increases as well. In the images below we'll use a stepover equal to 1/10, 1/5, and 1/3 of the tool diameter to show this correlation. To put real numbers on this, that would be equavalent to a .012, .025, and .042″ stepover for a .125″ ball mill.

Stepover =1/10 of diameter

Stepover =1/5 of diameter

Stepover =1/3 of diameter

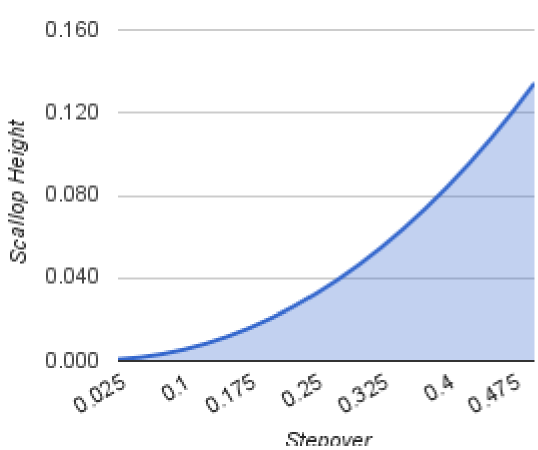
As you can see, the change in quality is so dramatic that you might be tempted to always use the smallest stepover possible.

Speed vs Quality

It shouldn't be surprising that you'll have to give something up if you want to use a really small stepover. In this case you'll trade time for quality- you give up machining speed to use a small stepover or give up quality if you want a quick machining time. This is easy to understand when you consider that the total length of a toolpath will approximately double if you cut the stepover in half. The question is, "Will cutting the stepover in half double the quality of my part?"

The Sweet Spot

It turns out that there is a point of diminishing returns in the time/quality tradeoff. Below is a graph of scallop height vs stepover that illustrates the effect. The graph has been normalized to a tool diameter of 1.0 so it's easy to scale it to any tool you happen to be using. (Click on it to see a bigger version)

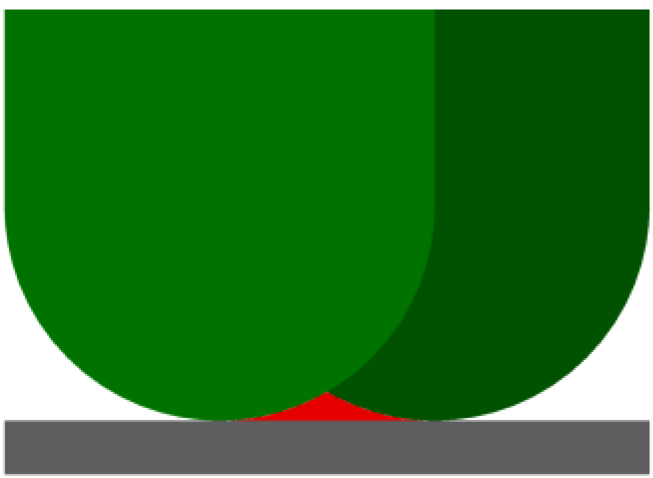


Scallop vs. tool diameter

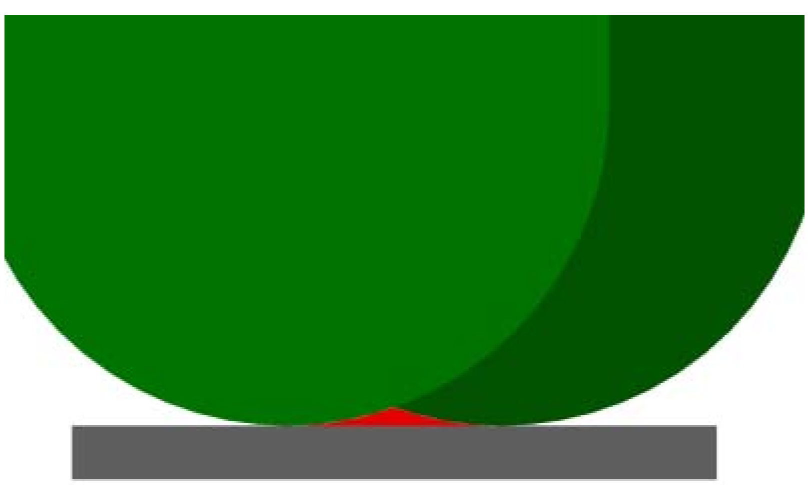
The important thing to note is the shape of the graph- it tends to flatten out when the stepover goes below about one eighth of the diameter. This means that when you go below this point you're going to take more time to machine without a proportional gain in finish quality. If you're machining a steel injection mold then it may still be worth it but you really need to be sure before doing that.

Scallop vs. Tool Diameter

Here's the other thing we can glean from the math behind the chart above- for a given stepover, a larger tool will give you a smaller scallop. This means that you can get a better finish "for free" if you can use a larger tool. Obviously, this only works if a bigger tool will fit into all of the parts of your geometry but this is one of the few "win-win" things we can do get better results if it does work for your geometry.



Scallop with a small tool



Note the reduced scallop when a bigger tool is used even though the stepover is constant.

Keep the Material in Mind

Before you figure out what stepover you need to get a .0001″ scallop, think about what you are going to machine- wood, tooling board, aluminum, steel, etc. I can tell you that in many cases you can do 10 minutes of sanding on a wood part to get a finish that would have taken you an extra hour or two to get straight from the mill. Likewise, tooling board like Renshape can be hand finished quickly enough that it may not be worth doubling the machining time to get a better finish. If you're cutting steel or other hard materials then it's probably worth letting the mill do more of the hard work.

The second characteristic of the material to consider is what kind of detail it can hold. MDF will not hold features in the .01″ range but metal will. If your material cannot hold a detail that is smaller than your scallop height then you do not need to reduce the stepover; doing so will only waste your time without producing a better finish.

Keep the CNC Machine in Mind

It may be a poor craftsman that blames his tools but we do have to be realistic about the nature of our equipment. In particular, how long do you trust your mill or router to run trouble-free? I started out with a small table-top mill that, while very good, could not be trusted to run for hours without missing a step or hiccuping in some way that gouged a part I had waited half a day to get. If you have a machine like this then it's worth thinking about the picking the maximum stepover based more on machining time than finish.

Rules of Thumb

That was a nice bunch of pictures but you may still be left with the question, "So what stepover do I use?" Here are a few suggestions:

- The stepover should be between 1/3 and 1/10 of the tool diameter
- Use a larger stepover, in the 1/5 to 1/3 range, for soft materials that cannot hold detail well
- Use a smaller stepover, in the 1/5 to 1/10 range, for hard materials or materials that can hold significant detail like metal and jewelers wax
- Use the largest tool that will allow you to machine your geometry

Once you have a few projects complete you can adjust the guidelines above to suit you materials and machine.

[Speeds and Feeds]

Posted by [Robert] on May 16, 2011 in [How To] | [0 comments]

I've been planning on doing a writeup on feedrates for beginners but it looks like Bob at [CNC Cookbook] beat me to it. His article goes way beyond the beginner version so hopefully everyone can benefit from it at some level.

Here it is [CNC Milling Feeds and Speeds Cookbook]

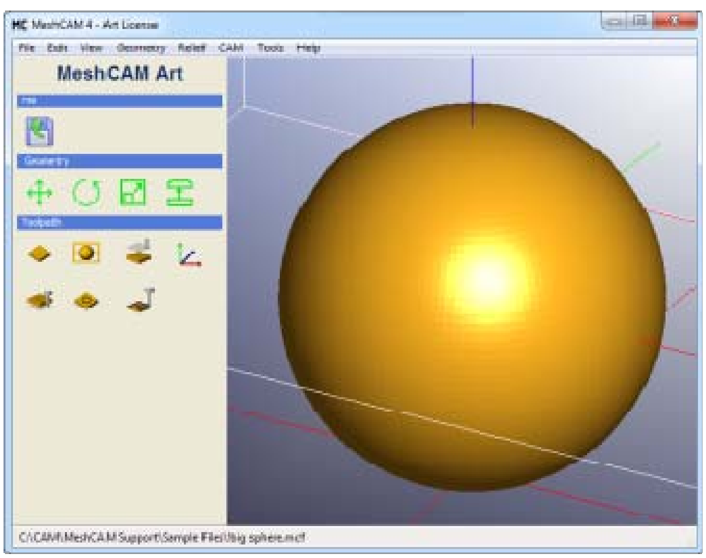
[Surface Angle Limits]

Posted by [Robert] on May 10, 2011 in [How To] | [0 comments]

I get two questions over and over: "What is the surface angle limit?" and "Parallel finish leaves ragged edges, what can I do?" Each question is really the answer to the other but it's something I've had a lot of trouble communicating. Here's my latest attempt complete with pictures…

A Simple Geometry

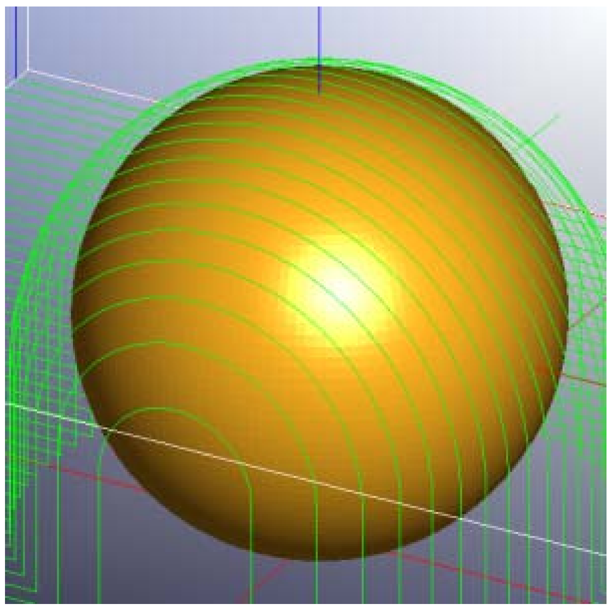
Below is a simple geometry that I've loaded into MeshCAM:



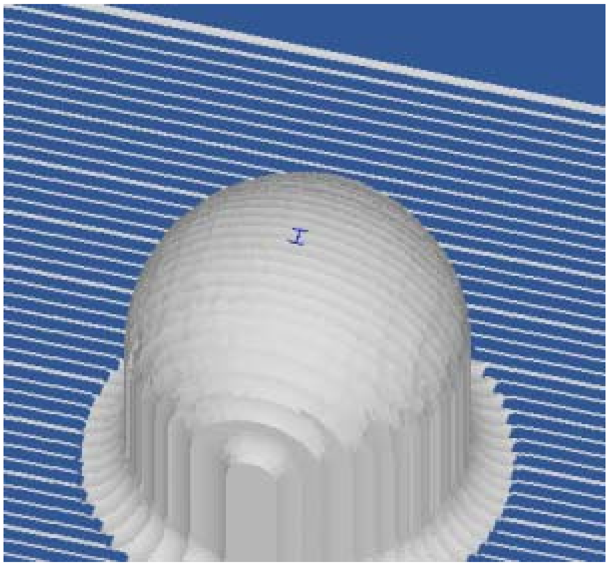
As you can see, it's a very simple sphere but it has two very important characteristics that can be used to illustrate the benefits of the surface angle limit function, it has a relatively flat area on the top of the sphere and vertical walls at the edge when machined on a 3-axis mill. All toolpaths shown below will use a .25″ ball mill with a .05″ stepover and stepdown.

Unconstrained Parallel Finishing

Below is what the toolpath looks like if we just use the parallel finish option:



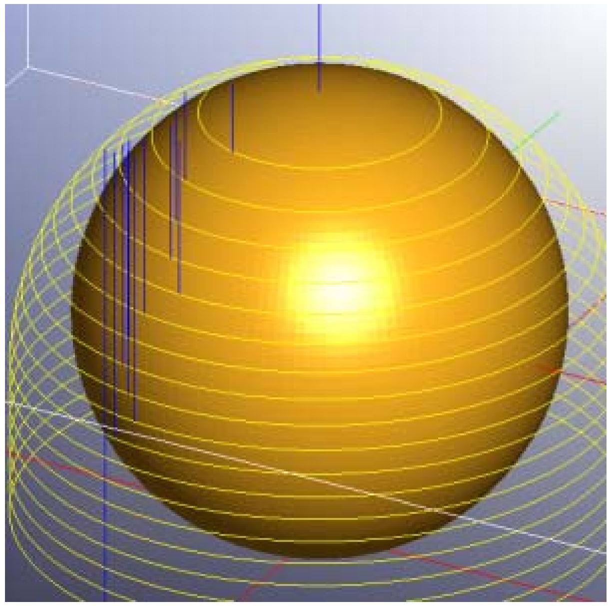
This is probably a very familiar-looking toolpath if you've spent any time with MeshCAM. If we simulate it in Cutviewer we get the following:

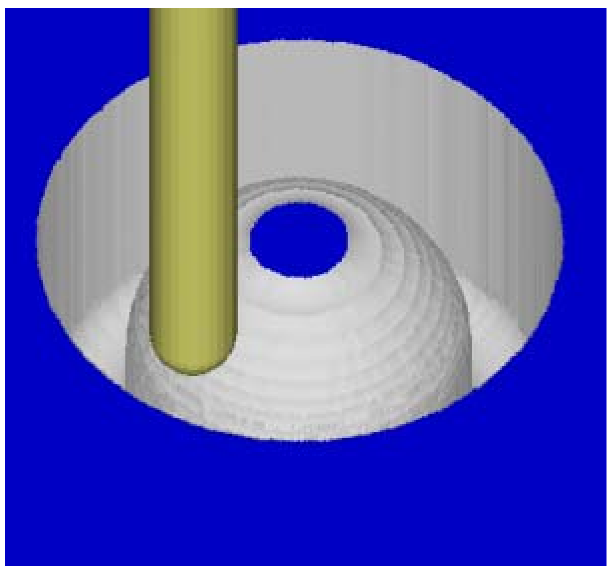


Note that the vertical edges are really rough while the top is relatively smooth. This illustrates the most important characteristic of parallel finishing: you get good results for shallow areas and poor results for steep areas. We could get a better finish on the whole part by reducing the stepover but that has a direct impact on machine time and machine time is something that most people strive to reduce.

Waterline Toolpaths

Waterline toolpaths are in many ways the exact opposite of parallel paths- you want them for the steep areas and want to avoid them for shallow areas. Here is the same part with a waterline toolpath:



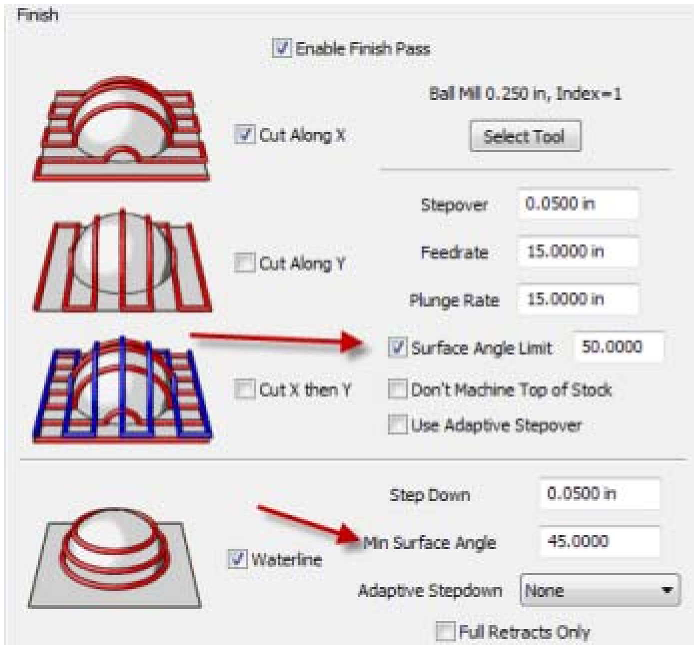


Note that the simulation shows the opposite of the parallel finish- the shallow top of the sphere is really rough and the steeper parts look good. As in the parallel, we can reduce the stepdown to get a better overall finish at the expense of machining time but that is never something to consider if you have other options.

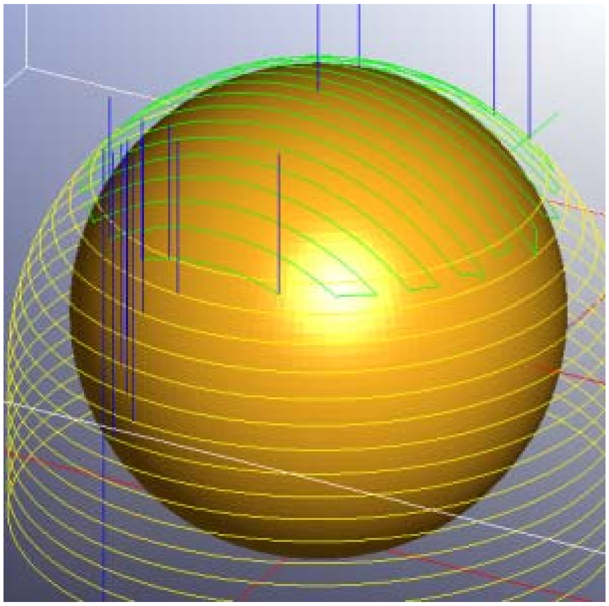
It's also easy enough to enable both parallel and waterline toolpaths at the same time to get a better finish but that is like reducing the stepover/stepdown- you are just trading the finish for machining time since musch of the geometry would be machine twice. Surely we can do better.

Combining Parallel and Waterline

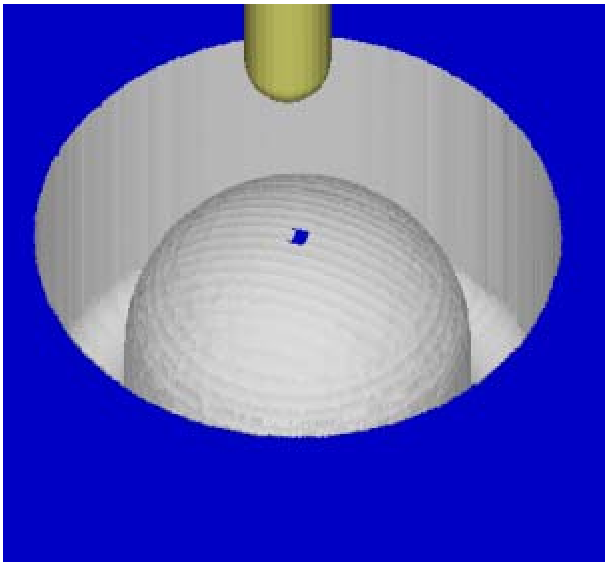
This brings us to the point of this post, the "Surface angle limit" and "Min surface angle" settings:



The purpose of these setting is to tell MeshCAM where to apply the parallel and waterline toolpaths. If the "Surface Angle Limit" parameter is enabled, only areas that are flatter than the defined angle will be machined with the parallel path. Likewise, only areas that are steeper than the "Min Surface Angle" value will be cut with the waterline toolpath. Generally I like to make them overlap slightly so there is no hard edge between the two in the finished part. Using the values above the toolpath looks like this:



That get's us close to an ideal condition- steep areas with waterline, shallow areas with parallel and most of the geometry is machine only once. That's as close to win-win as you can get.



The Short Version

Here's a quick recap of why surface angle limits are such an important feature:

- Machine steep areas with waterline
- Machine shallow areas with parallel
- Only machine each part of the model once with no overlap
- Get the best finish possible without extending the machining time

For many users this should be something that is used on every job.