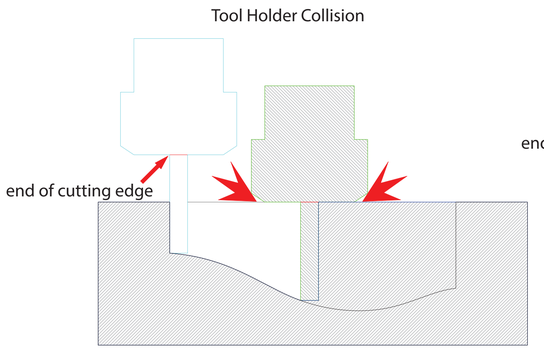
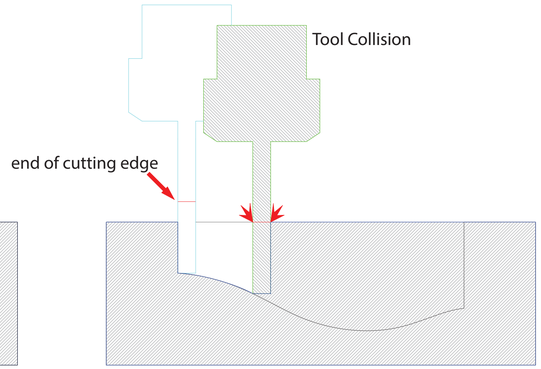
CNC CONCEPTS AND EXPLANATORY IMAGES

Tooling Collisions



The tool holder strikes the material or stock. This causes damage to the machine and the material.

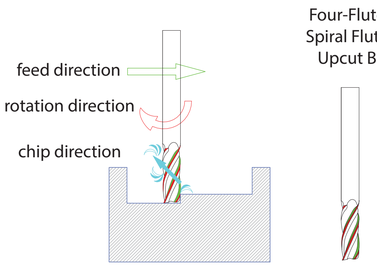
This cannot be allowed to occur in any case!



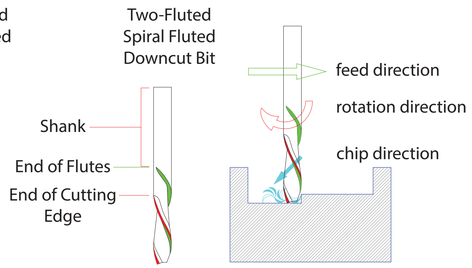
The bit is engaged in the material beyond the end of the sharpened portion of the bit. This generates additional friction damaging the tooling or shortening tool life. It also creates extra wear and tear on the CNC machine because it is doing more work.

Upcut vs. Downcut Bits: How they work

Note: Maximum cut depth per pass is 50% bit diameter

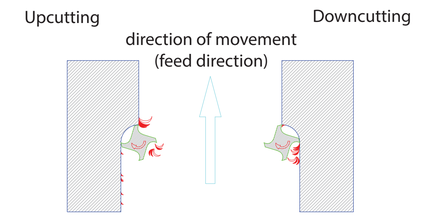


This type of bit often results in a feathery top edge but it reduces strain on the equipment and reduces chip and heat buildup. It will put upward pressure (lift) on the stock/materials.



This type of bit often results in clean top edge but it increases strain on the equipment and pushes chips down into the cut. It can result in increased heat due to friction. It always presses down on the part.

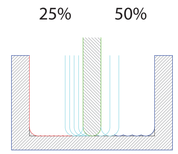
Upcut vs. Downcut relative to Feed Direction



Unsupported fibers tear away so the edge looks rough and fibrous, but the cutting edge pulls the bit into the piece, reducing chatter and bit deflection.

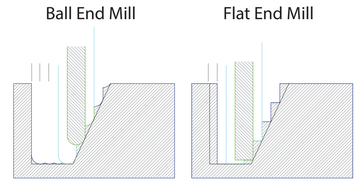
Fibers being cut are generally supported so the edge cuts cleaner. The forces on the cutting edge make the bit want to deflect more and can cause chatter and wear.

Comparing Stepover Percentages



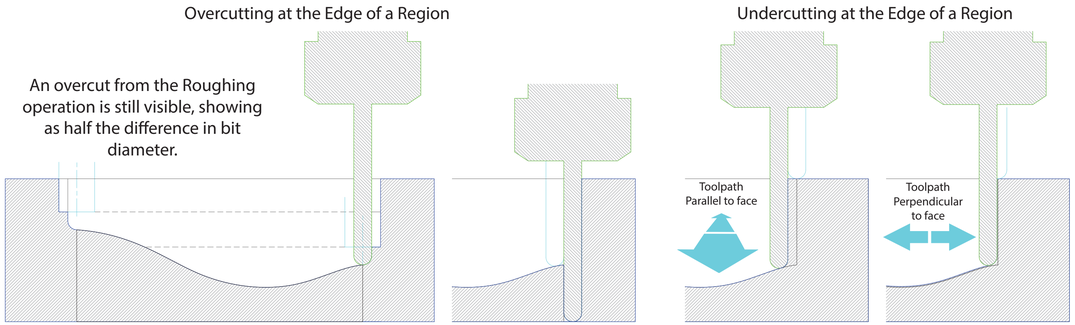
We prefer to define this as a percentage so it adjusts automatically if you change the bit. The maximum stepover is usually 50% of the bit diameter. Decreasing the stepover refines the finish of a part, but drastically increases the cut times.

Stepover on a Steep Slope



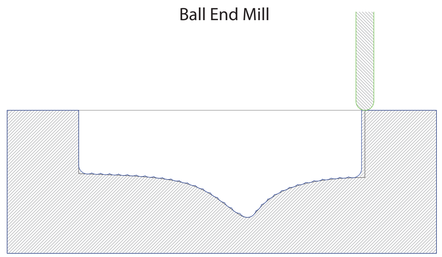
When finishing a part, the bit choice and the toolpathing direction relative to part geometry will affect the result. Steep slope resolve better when your toolpaths perpendicular to the sloped face. If you run parallel you will have large ridges that can be made worse if you are using a flat end mill. The stepover increment can also cause undercutting if it doesn't mesh well with the part dimensions.

Modeling a Frame for Control

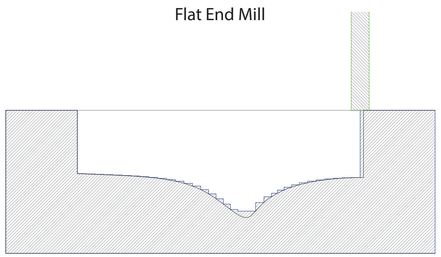


Modeling a frame around your part will prevent the software from overcutting, and possibly releasing your part; however, it may cause the software to undercut by as much as half the width of the bit instead.

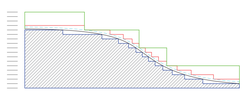
Stepover and Bit Geometry vs. Part Geometry



Bit choice is critical when finishing a part. We usually suggest a ball end mill because it does a much better job getting into irregular details and sloped valleys and crevices. If your part is primarily gently sloped, then using a flat end mill may be a good option because you may be able to do so and use a larger stepover to decrease your milling time without greatly affecting the part. However, a flat end mill will square/flatten out any crevices. Using a hybrid approach by dividing up your part into regions for each bit is another option, but that will likely require more time spent on tool changes.



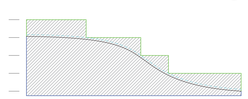
Laser Contouring vs. Horizontal Roughing



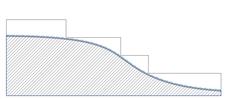
Laser cut contour models are cut so that the top edge of each model layer represents the height of the slope. While Horizontal Roughing leaves a similar looking result, its peaks are much higher than the original slope. If used as is, these models will not demonstrate correct height relationships to the rest of your model.

Horizontal Roughing and Finishing

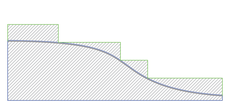
Roughing and Finishing with Normal Tolerances



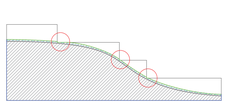
The roughing pass uses a larger tolerance and leaves a suitable space between its low points and the part for the finishing to clean away.



If the tolerances are switched, or the bit touch off isn't done...

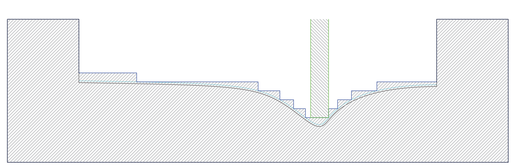


The roughing pass cuts too close to the part and the lowest parts of the operation remain after finishing, leaving gouge-like tracks.



Horizontal Roughing vs. Horizontal Finishing

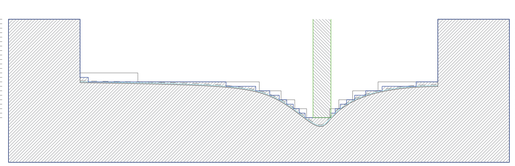
Horizontal Roughing



Horizontal Roughing can be used to prep a part for either Parallel Finishing (most common) or Horizontal finishing. It is usually done with the largest bit that makes sense to save time. Its purpose is to remove the bulk of the waste material to reduce the strain on the bit used for the finishing pass which is usually much smaller so you can produce a part with a finer finish. It removes an entire level of material and stays a set distance from the part.

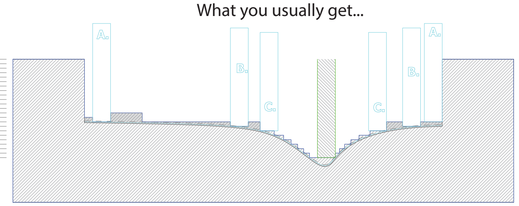
Horizontal Finishing

What you expect...



The result that is usually expected from the Horizontal Finishing operation is something similar to the roughing operation, but with a much finer step down, and a resulting finer resolution.

What you usually get...



Horizontal Finishing, by default, is defined solely by vertical step down increments. It usually only makes one or two passes at each level that is defined by the programmed step down. As a result, areas that have a shallow slope may be ignored after a perimeter pass at that level. Adding a "Horizontal Override" to the operation will introduce a step over definition that addresses this but will also drastically drive up file size and calculation, simulation, and milling times.