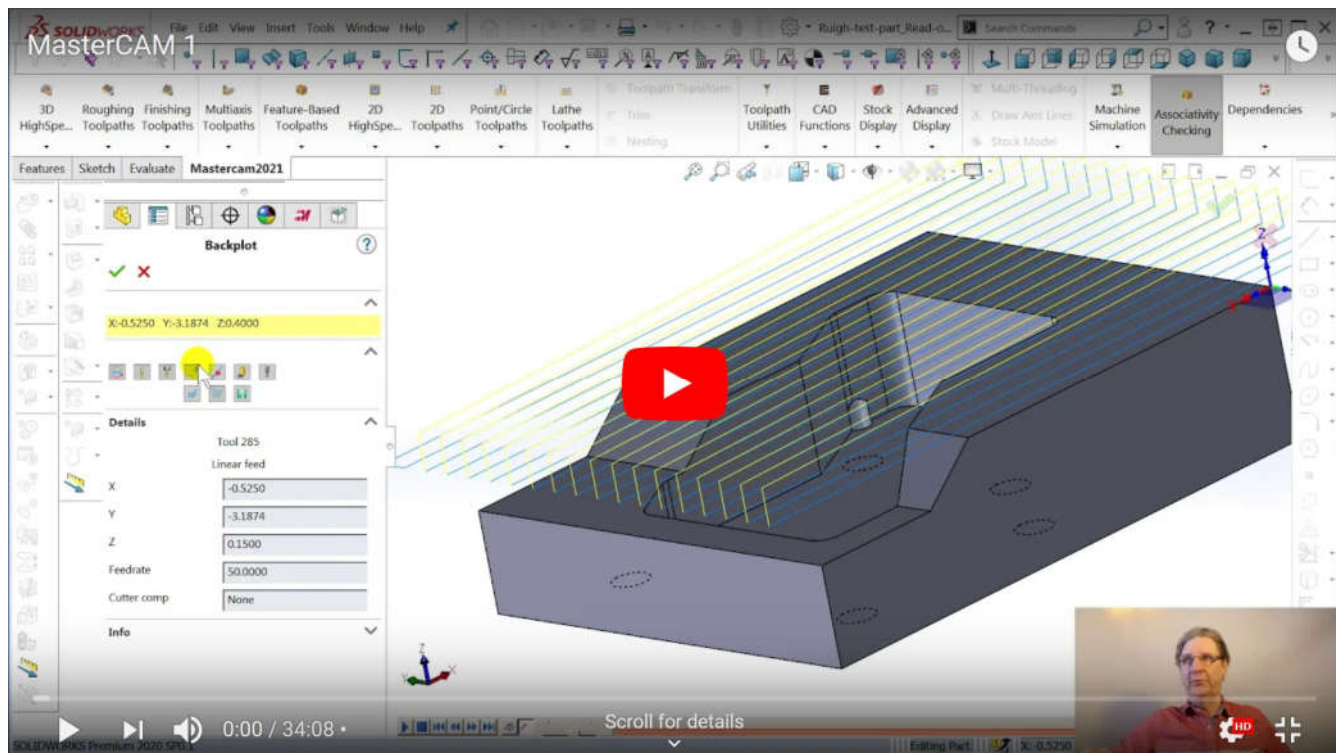—RAKO STUDIOS—

Rako Studios » Media » Suffering-with-software » MasterCAM second impression

MasterCAM second impression

I manage to make two pocketing operations, but it seems the program wants to fight me at every turn.

Beginning part file here.
Finish part file here.

My second impression of MasterCAM is not much better than the first. It makes great toolpaths, but the user interface is so complicated, it took me a half-hour to do two pockets, and that was after rehearsing for hours. Unlike SolidWorks CAM by CAMWorks, VisualMill, or BobCAD CAM, MasterCAM could not take the top edges of the pocket as a geometry since they were not already on a single plane. The other programs did not care, they knew my design intent.

This meant I had to make a sketch to do the deep pocket. I looked at using the "CAD Tools" button in the Command Manager, but that just wasted 30 minutes of my life. It can't project the side faces to the construction plane, nor can it project simple edges to the plane. No matter, I prefer SolidWorks sketch tools anyway. I re-selected the geometry to the new SolidWorks sketch and then did the normal fiddling with the "Parameters" tab choices. It made a beautiful toolpath, spiraling into the pocket and using that high-zoot adaptive machining, sometimes called high-speed machining, VoluMill, iMachining or HSM.

It looked like I was ahead of schedule as I went to make the shallow pocket that defines the little plateau in the bottom of the pocket. This progress turned out to be an illusion. I did learn that the Toolplane seems to define where the toolpath starts cutting. I remembered how the depth of cut is in the "Linking Parameters" tab of the Parameters dialog box, if only since that was so confusing.

I had to draw a SolidWorks sketch to define the lower pocket, but I had to do that in the other CAM programs as well. This is because there are two holes that come up from underneath, one through and one breakout-- partially in the pocket and partially blind.

Things were going well, since I remembered all the settings and dialogs from the earlier pocket. Just when I thought I was done, I noted that this lower shallow pocket was not accurate, it left a lot of material in the corners, and did not cut a little side channel that was wider than the 1/4-inch endmill.

It took a long time to find that there was a default setting for "Corner smoothing radius" of 0.1, and that is why the toolpath was not cutting the true extent of the pocket. This setting was in the "Trochoidal cuts" section of the "Roughing" tab of the Parameters dialog box. Note that this section was marked to "Off". Why a roughing parameter would affect the final cut, or why a section marked "Off" would matter, or what a trochoidal cut is, or why this was defaulted to a value that made a bad toolpath is a mystery to me.

Equally mysterious is the program's love for planes. The Tool Plane, the Construction Plane, the Work Coordinate System (WCS). My pal told me he thinks the WCS is what sets part zero. I learned that the Tool Plane will set where the toolpath begins.

I had to make two sketches in this program. Watching some tutorials of the stand-alone MasterCAM that used their method of projecting geometry to a plane convinced me I should stick to CAM programs that work inside SolidWorks.

It's not only the need for CAD when doing CAM. A SolidWorks add-in CAM system means that you won't have multiple files floating around that you are never sure are at the same rev level.

Most of the internal programs brag how changing the part will "automatically" adjust the toolpaths. I found this a fiction. In SolidWorks CAM it want to re-recognize features and just apply its Technology Database, even after you spend hours to change what the automatic feature recognition did. I would much prefer to explicitly go and modify the toolpaths myself, so I know exactly what is going on.

Automatic sounds nice, but it also means "unknowable" so it will be easy to assume the program did the right thing, when really it did not do what you expected. And so far, none of the CAM programs seem to do what I expect.

There are a few "Gold partner" Solidworks CAM programs I don't plan on looking at. OptiCAM is for optics, CG CAM Tool is 5-axis mold-making stuff, PathFinder3D is for show displays. I would like to look at HyperMill but they don't want to let me have a demo program since "I am a novice user". OK, fine. I will spend my money elsewhere.

I will keep at these MasterCAM videos until I finish the part. I would not be confident to put the G-code into a machine, since I still don't know what sets part-zero, or how different toolplanes will affect the G-code.