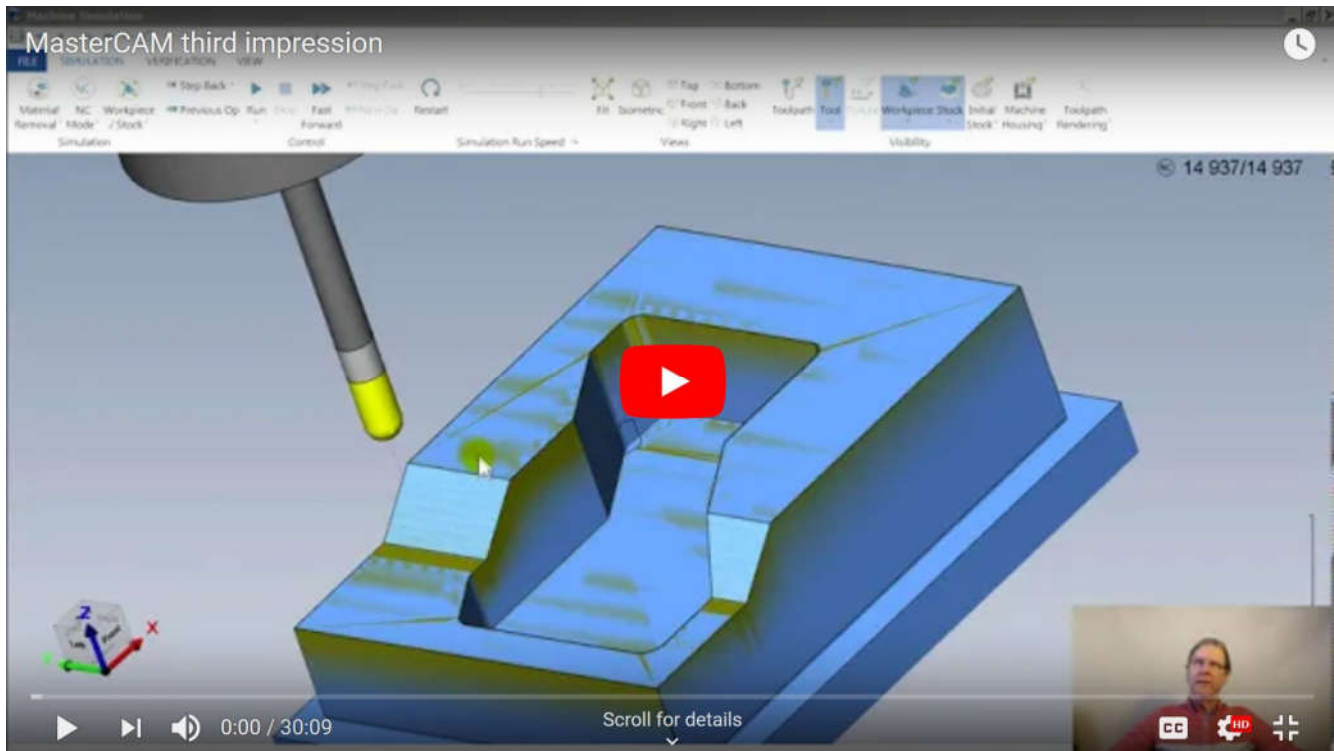




Rako Studios » Media » Suffering-with-software » MasterCAM fourth impression

## MasterCAM fourth impression

**My machinist buddy shows me the right way to do some of the toolpaths. I start on the bottom of the part.**



Beginning part file here.  
 Finished part file here.  
 My buddy's part here.  
 Parts will only open with toolpaths in the demo MasterCAM for SolidWorks program.

When I asked my machinist buddy Dave about doing setups and part-zero, he was nice enough to take pity on me and send me the part he designed with MasterCAM toolpaths. Since he designed the part to trip up CAM programs, he knew how to deal with some of the difficulties. He used to sell MasterCAM, so using it is in his comfort zone. He went so far as to upgrade his personal SolidWorks to 2020, and he downloaded the same demo program I have from MasterCAM.

While I managed to get the top facing and the perimeter operations good enough, I noted Dave did not need a sketch to drive the deep pocket. That is because he knew there is a "project to toolplane" button below the pick box in the geometry dialog panel. That let me get rid of one sketch.

Next, for the shallow pocket, he showed how it is possible to climb up and down face edges to define the lower shallow pocket without a sketch. The tool will try to enter into the breakout hole, but since it can't fit, MasterCAM knows not to barge through the stock. It does put a little "divot" in the toolpath, but that should not be a problem, other than appearance on the pocket floor.

Doing this let me delete the sketch for the shallow pocket. Dave did need a sketch to do the flat area next to the slopes. I drew a rectangle and tried to define an open pocket. Dave drew a line where the slopes meet the flat and then did a contour operation with enough passes to do the whole flat.

Up till then, my toolpaths worked fine, they just were not as elegant, or I made sketches that I did not have to. What really impressed me was how Dave cut the two slopes.

I used a ball end mill. MasterCAM is really serious about making the ball paint over the slope, so that it did not leave a little cusp of stock on the left edge of the smaller slope. Most other programs did this, although BobCAD CAM was able to overcome it by selecting "3D extents" as an option.

The problem came at the bottom of the slopes. In order to do the slope surface right up to the lower edge, the toolpath drove the cutter into the flat. The put two 0.020" gouges along the edges where the slopes meet the flat.

What Dave did was genius. He dispensed with the ball mill and left the square end mill in the spindle. Then, instead of rasterizing across the part from side to side, he came down the slopes. When the mill reaches the bottom, the flat end cannot gouge the material, yet there will be a nice sharp corner where the slope meets the flat.

I did not rough out the material first, so maybe my toolpath was too aggressive. Still, the principle is what counts. I can see just doing another toolpath that does go across the sloped faces but leave a staircase of stock running down the slopes. Then the finish toolpath could run down the slopes and leave a great finish, probably better than a ball-mill.

What I really love about this is that it saves me a tool change to the ball mill, and a tool change back when I go to do the bottom of the part. My Avid Benchtop Pro does not have an automatic tool changer (ATC) so it would take me many minutes to change tools, find the tool height offset, and start the operation.



With the top operations all fixed up, I went to do the bottom of the part. I am pretty sure I got part-zero defined right, set to the bottom corner of the part. This would have to mean finding the Z height of the parallels that the part is sitting on. I assume I could have a part-stop setup so that I would not have to find X- or Y-zero again, I could use the previous ones.

It took some fiddling with the planes, It did not occur to me I could not have the plane be the bottom of the part, and the origin at the top, up against the vise parallels. That meant some frolicking around in the "Linking parameters" where you set the depth of cut.

I still do not feel confident with what MasterCAM is doing, and I wonder if I would be able to post good g-code that acted like I expected. Next episode will be putting in the five holes, and like the other programs, anything "automatic" will not work right.