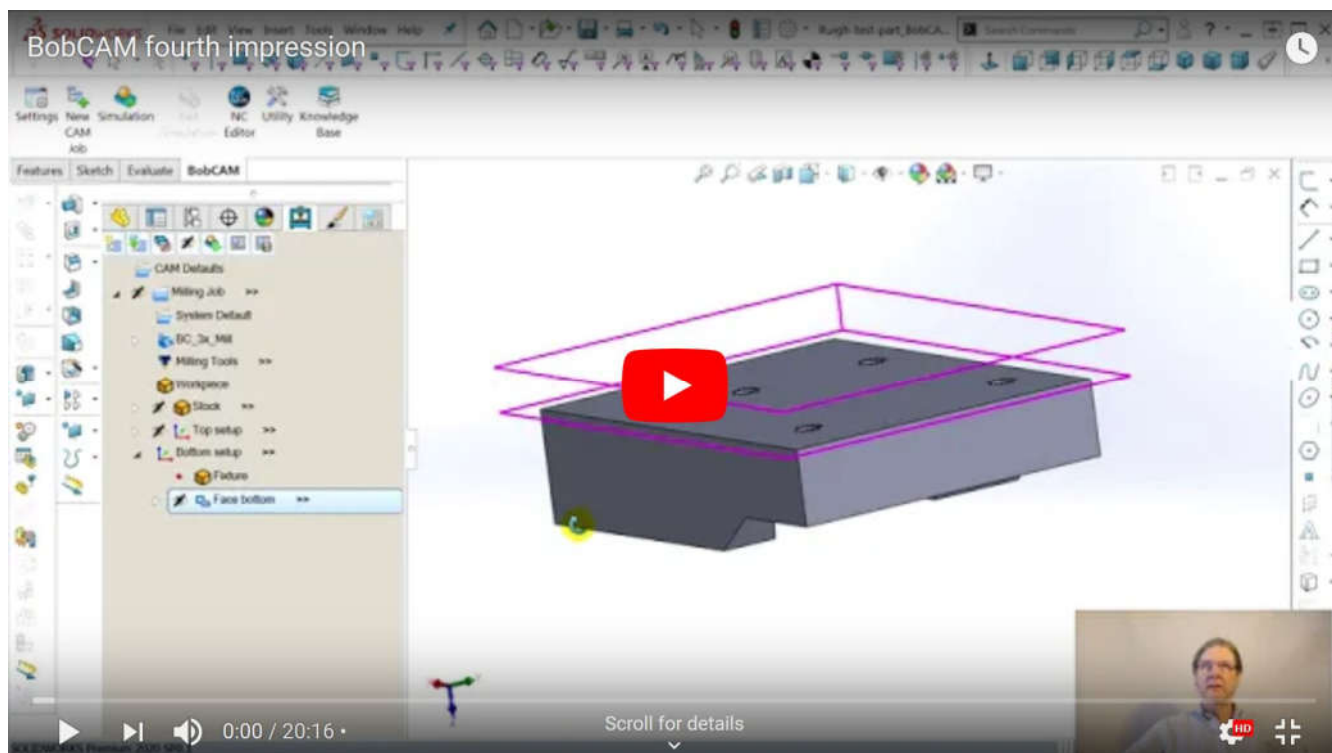




Rako Studios » Media » Suffering-with-software » BobCAD-CAM fourth impression

BobCAD-CAM fourth impression

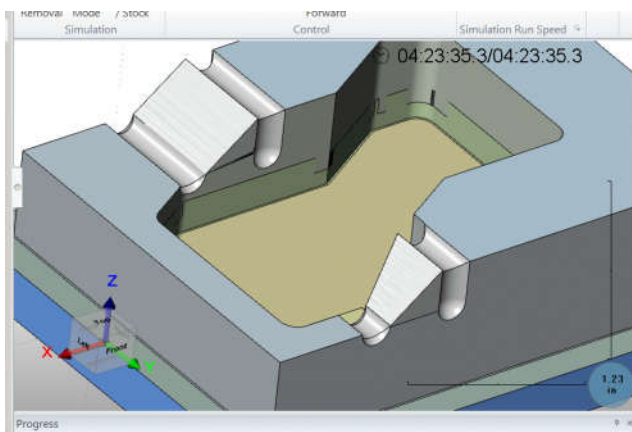
A fourth session with BobCAD CAM gets the part finished. It still needed some help with the tapped holes.



Beginning part file here.
Finish part file here.

Note the demo part file will only open with toolpaths in another demo version, you can't open it in a paid-for BobCAD CAM.

I figured out how to make the slopes without a little gouge on the top. Instead of "Advanced Z-level" I used "Advanced Planar". This made nice tight toolpaths with a minimum of machining time. I never did figure out what was putting the curlicue in the Advanced-Z toolpath. Probably operator error. I am astonished with how CAM programs will



machine the through the solid model. This is what happens if you select the "Bottom" option in one of the 3D machining operations.

Setting up part-zero in the bottom was straightforward. I selected a new origin, referenced to the part this time instead of the stock. If the bottom part-zero was on the stock, then the stock thickness would change the part thickness.

With the origin selected, it was typical stuff to align the co-ordinate system the way I wanted to set the part into the mill. This is with Y-positive going back and X-positive towards the right. With the part-zero origin set, I could then turn to facing off the bottom and putting in the five 1/4-20 tapped holes.

Doing the facing was as easy as doing the top face. The secret is not to select any geometry. Then the program assumes you want to machine across all the stock, not just pocket out the bottom face of the part. I learned MasterCAM works this way, so that might explain why SolidWorks CAM and VisualMill do the same.

The program defaulted to doing the 0.4" thickness in a single depth. With my Avid BenchTop Pro, I thought it would be better to take gentler cuts, 0.1" deep.



The Avid Benchtop Pro is a router-style mill.

All this fighting the program is making me. While I am still not fond of the simulator in BobCAD CAM, I will admit it was my watching the toolpaths in the simulator that made me decide to handle the facing in multiple depth cuts.

Doing the five tapped holes required a bit of experimentation. I first tried to select the holes by just selecting the bottom face, as this works in VisualMill. The program does not recognize any holes when doing this.

Next, I selected the cylindrical faces of the holes. The program saw that two holes were through-holes, and put them in one group, and then made the other three blind holes in another group. In once rehearsal, it also broke up the operations, doing all five center drills at once, but then doing a drill, chamfer, and tap for 2 holes and another drill, chamfer, and tap for the 3 remaining holes.

You can go into an operation sequence dialog and re-arrange them so all the drills, chamfers, and taps happen in a batch of five. This time the program combined the operations on its own, doing 5 center drills, 5 drills, 5 chamfers, and 5 tapping operations.

To get rid of the two separate groups in the operations tree, I started over by selecting just the edges of the holes. BobCAD CAM put them all in the same operation group, but made the holes 1.5" deep. I changed this to 0.38, the actual depth of the holes, and BobCAD CAM did all the operations properly.

The simulator showed the holes as gouges like SolidWorks CAM did. This is because the model the tap as 0.250 and the hole is modeled as 0.201. When the tap runs though, the program thinks it did a gouge, but this is the way it has to be, due to modeling convention.