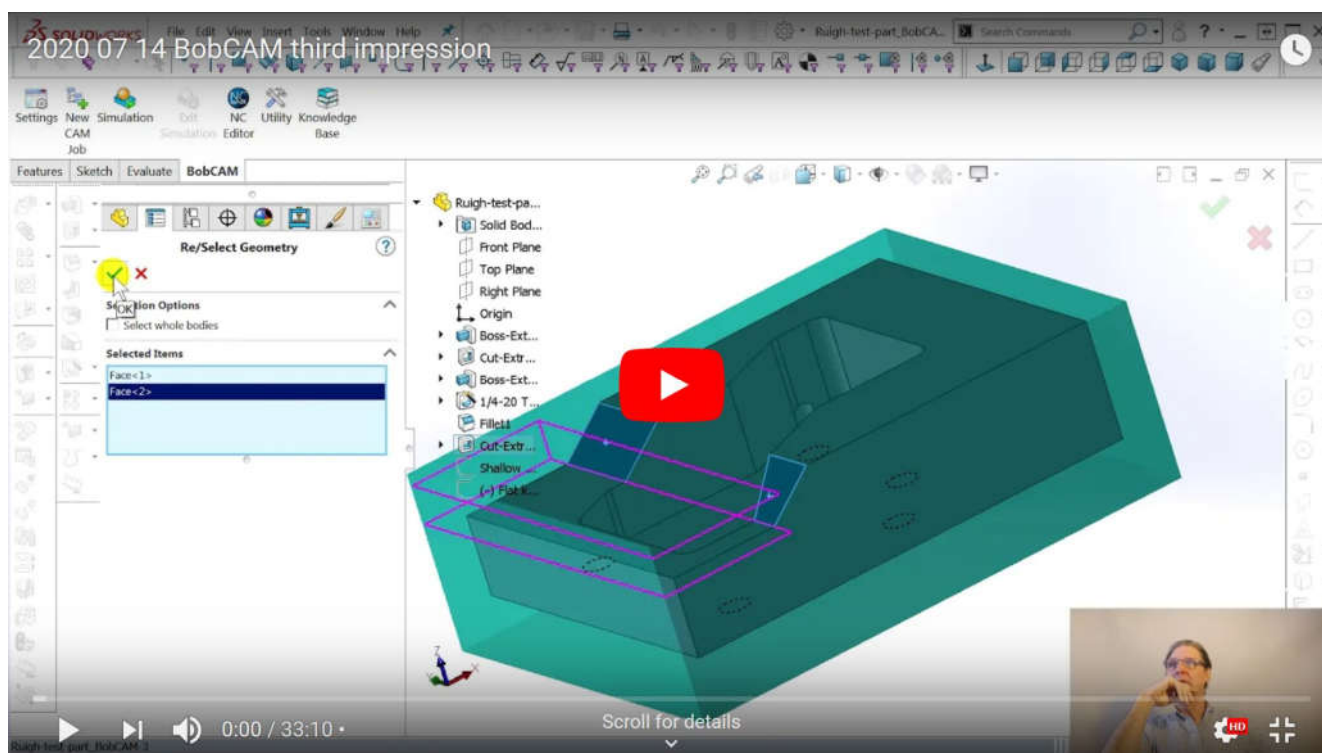




Rako Studios » Media » Suffering-with-software » BobCAD-CAM third impression (youtu.be/XXuXXByJAGQ)

## BobCAD-CAM third impression (youtu.be/XXuXXByJAGQ)

A third session with BobCAD CAM gets the top of the part finished. It still needed another sketch, and it left a gouge.



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.

Getting used to BobCAD CAM, I managed to mill the flat area and the slopes. Like the other CAM packages, I could find no simple way to finish the flat, I had to draw a sketch bigger than the face and then BobCAD CAM thinks it is a pocket and removes the material in the corners. The slopes I did with a 3D toolpath, and it almost worked perfect, until it didn't.

BobCAD CAM put a little curlicue on the toolpath between the two slopes, and that put a little gouge in the top of the part. Worse yet the simulator gouge verification report did not pick it up.

I tried entry and exit parameters but nothing seemed to change this screwy behavior. I fear the solution will be to make yet another sketch on top of the slopes and use that as a geometry to run the toolpaths. BobCAD got so close, unlike the CAMWorks and VisualMill, it machined the faces well simply by invoking "3D extents" in one of the multitude of dialog boxes that all these CAM programs love.

Machining the flat area next to the slopes was a bit frustrating. I thought I could just say to face this face, and BobCAD would just mill over the face I defined as geometry. Instead, it cut the entire part in half.

So I went back to SolidWorks and drew a rectangle over the flat area, but bigger at the edges so that the pocketing operation would not leave the corners undone. The fact I have to let BobCAD think it is a pocket means that the tool will plunge down inside the pocket to get its depth of cut, rather than come from the outside.

I tried to mess with that, but I think the best I could do would be to fool the tool into plunging in the overhang area where the deep pocket is already cut. That is not a general solution so that bothers me.

I am not a machinist, so all this might be operator error on my part. When I look at other programs, many have an operation called "open pocket" and that is what I think this should be handled as. When I try to define it that way in those programs, the toolpaths generated went all the way around the part and into the deep pocket. These areas were already cut, so I need to learn a way to define a keep-out so the tool does not wander all over.

VisualMill let me extend the toolpaths on a 3D finishing operation, so maybe that would be one way to fool a CAM program into doing this flat with the minimum of wasted time. Despite all the feature-recognition hoopla, I think I will be spending a lot of time doing sketches to force the toolpaths to do what I want.

I just wish the simulator was better in BobCAD CAM. Those little text boxes it litters all over the part are far less useful than the color-coded gouge and excess representations in other programs.

All this fighting the program is making me realize why the software companies talk about their technology database or their knowledge base. That is another way of saying you have to spend an immense amount of time doing clerical work to tell the program your preferences.

Thing is, I don't think my preferences are so screwy that they need to be custom. BobCAD only lets you select a geometry and then you set "System Compensation" to decide whether you are cutting a pocket or a perimeter.

I prefer the companies that let me say what I am trying to do, so that they can make better default behavior. If I say I want a rough pocket, it should clear out all the material in the pocket.

If I want a rough perimeter, the program should know that stock will have little pillars left on the corners unless the toolpaths go mill those off.

Similarity, an open pocket might have some chance of doing the flat on this part the way I want.

I think about my machinist buddies, and their mentality seems to be, "How will I hold this thing as I machine it?" That means that these programs should have smart provisions for vices and clamps.

Then my buddies say, "What is the biggest tool I can hog off the most metal the fastest?" And I would like to see a program with that thinking. I believe VisualMill did do something like that-- though it had a 2-hour machining time, that was with a 1/4-inch endmill.

And I really need a program with a good simulator, that simulates the G-code, not the internal math. That gives me some real trust.