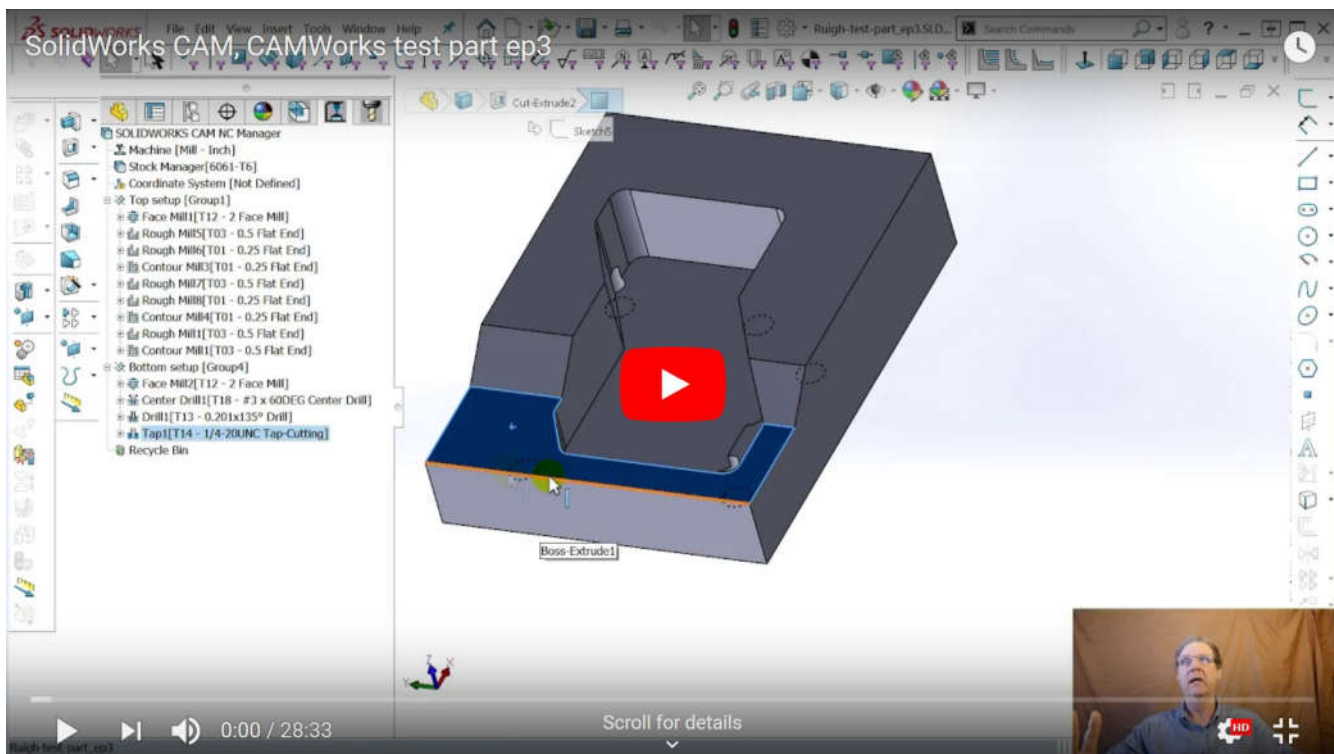




Rako Studios » Media » Suffering-with-software » Solidworks CAM, CAMWorks test part ep3 (youtu.be/cVmFvRcjp08)

## Solidworks CAM, CAMWorks test part ep3 (youtu.be/cVmFvRcjp08)

SolidWorks 2018 and later has a free CAM (computer-aided manufacturing) add-in to make G-code for milling machines.



Original part auto-recognized here.  
 Test part after pockets in ep1 here.  
 Test part after holes in this ep2 here.  
 Test part after perimeter & 3D this show here.

In this episode I take the test part made by my pal Dave Ruigh and add a perimeter cut around the part. When Solidworks CAM / CAMWorks did this automatically it ran the cut down to the bottom of the stock, which would have crashed into the vise. Here I extended the cut 0.020 below the part, so it would not leave a sharp edge after facing.

While it is obvious that automatic anything does not work well in this, or any CAM program, I am beginning to like the excellent simulations that SolidWorks CAM does. It does seem a bit clumsy, but I am a new user and things often feel awkward with a new tool.

After getting the perimeter done, I did the sloped surfaces using the "Multi--Surface" feature recognition in SolidWorks CAM. This added an area clearance and a Z-level operation that made the slant. The clearance operation was absurd, I have no idea what I

was supposed to set or do to keep the tool from hogging through the entire part and running around the outside of the part. I simply deleted it. I am sure a little reading or training and I would have better luck. Note that when Dave did a global feature recognition, it did nothing with these sloped areas.

What was nice was the set of traverses with a ball end mill that the program created to machine the slanted areas in the part. They were a bit course, with 0.050" steps down the slope, so I went crazy and changed the depth of cuts to 0.005". Because one of the slopes had a narrowing face as it went down, the program would pull up the end mill a little too soon. I could not find a way to extend the linear parts of the toolpaths to eliminate this, so I just made the entry radius much bigger, and that mostly got rid of the leftover material.

Ha, as I type this, I am watching a BobCAD video about 3D, and they make it trivial to extend the toolpaths, I must not be the only one that needs this, so I suspect I can do it in CAMWorks, I just did not see how.

What worries about this program is its complexity and dozens of dialog boxes with a handful of tabs each, all filled with either machinist jargon, or more likely, programmer jargon that they just invented to try and describe what the dialog was intended to do.

Programmer jargon is a problem in every CAD/CAM package I have ever used. It seems that we are expected to learn their jargon, things like database, and records, and "Z-axis minimum" instead of "depth". What I don't see is a programmer, or their boss going to work in a machine shop for a year, and learning to speak that language. All my buddies talk about "machine zero" and "part zero," but I have yet to see a CAM program use those terms.

Speaking of machine and part zero, that is one thing I should have done at the beginning. I had to understand how CAMWorks set these in the program in order to make my little angle iron part to test my Avid CNC Benchtop Pro. This is critical, since if you set it wrong, you could crash the tool into the bed or the vise or your hand.

This is one frustrating thing about SolidWorks CAM. I know I did it once, since I ran the mill up in the air over the vise and could see that it was doing both setups right. But even after figuring out how to do it once, I can not remember how I got there. Just some dialog boxes in some place and lots of clicks.

Right now what confuses me is that SolidWorks CAM wants you to set a "Coordinate system" but I am not sure how that affects the coordinate system of the various setups you create to machine a part on different sides of the part. I think I will post out the G-code for this part once I think I have it right, and see if it behaves like I think it should.

It does seem to pick up that overall coordinate system for the first setup. I have not verified this since I have not sent enough parts to my mill in order to see what happens. I have learned to hit that question mark help, as well as the help button on the bottom of dialog boxes.

That is another thing to like about SolidWorks CAM, the help system is pretty comprehensive. It has a little of that "reductionist" nature my programmer buddy used to complain about. "The feature normal checkbox changes the feature normal." Yeah, no kidding.

I purposely waded into this program without training or a lot of study to get a feel for how intuitive it is.