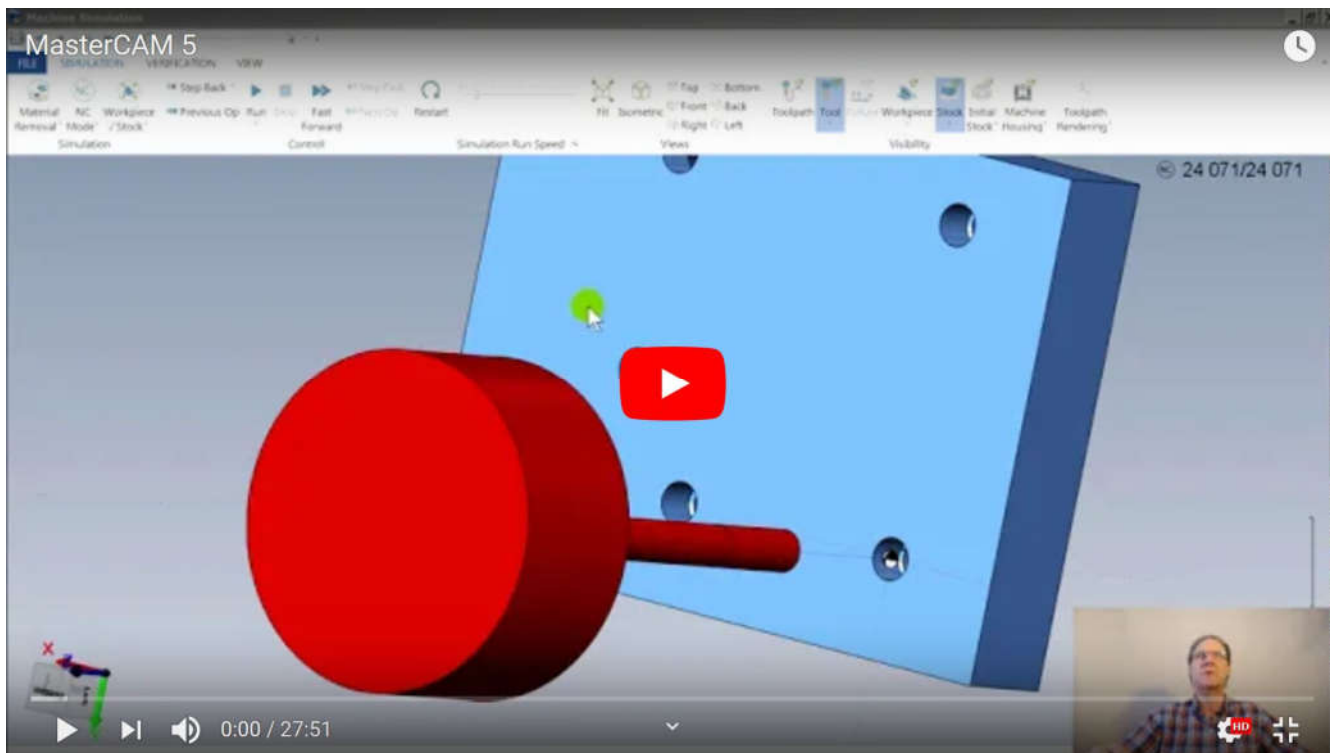




Rako Studios » Media » Suffering-with-software » MasterCAM fifth impression

MasterCAM fifth impression

Doing the five holes finishes the part. Naturally, automatic feature recognition does not work quite right.



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.

This last episode in the series sees me putting in the five holes from the bottom side. Since the MasterCAM demo program includes extra-cost options like feature-based machining (FBM), I thought I would give it a try.

SolidWorks CAM brags endlessly about being "feature based". With that program, it tried to do two of the holes from the top and three from the bottom. I guess this is what artificial intelligence would do, rather than genuine intelligence that drills the holes from one side.

SolidWorks CAM also insisted on breaking the holes into groups, even when I forced it to recognize all the holes from the bottom. It would make an operation for one through-hole, then a separate operation for the other through-hole, and then an operation for the three blind or partially blind holes. Then you had to rearrange things so it would do all the center drills, all the drill, and all the taps without a lot of needless tool changes.

MasterCAM also broke the automatic holes into groups, but in a different way. It did all 5 holes center drilling in one group. Then it did three tapping operations, the two through holes and the three blind ones. Then it did three peck drill operations.

Why on Earth it would do the tapping before the drilling astonished me. It is not too hard to move the drilling operations above the tapping, so that was an easy fix.

Just as concerning, and like the other programs, MasterCAM did not do the depths correctly. One tap was only 0.2" deep, despite SolidWorks defining them all in one group, and as .038" deep. This tells me that MasterCAM and the other programs do not use the SolidWorks hole feature data, they just look at the Boolean representation of the holes in the model, and decide what the operations should be.

MasterCAM also created a whole new toolgroup to hold these drilling and tapping operations. One downside for me was that with one center drill operation, three drill, three tap drills, the "FBM" container operation and the new toolgroup, there were nine line items added to the tree.

Like the other operations, it is very hard to see what operations are what features on your model, without tediously enabling and disabling the toolpath visibility, one by one. I now see why SolidWorks CAM by CAMWorks would light up the toolpaths as you hovered over the operations. This confused me at first. With one operation selected, that would illuminate those toolpaths, but then as your mouse hovered near that, other toolpaths would become visible then disappear as you move the mouse.

Now that I understand how easy this makes it to see what toolpaths are from what operations, I have a whole new respect for this method. I really dislike that MasterCAM puts any renaming text at the end of its long string description of the operations. That is usually off the panel. You can see what you named things if you hover over the name, and it expands.

It does not bother me that "Feature-based machining" did not work right. First, off, this is an extra-cost module in MasterCAM. More importantly, I prefer to do things by hand, individually, so I know what the program is supposed to be doing. It was a nice bonus here that nine line items got reduced to three.

I got those three operations by selecting the five holes for a drill operation, and assigning a center drill for the tool. MasterCAM chose a countersink tool as the center drill, or maybe it was trying to chamfer the hole entry for the tap. As always, a lot of fiddling with "Linking Parameters," which is what MasterCAM calls depth of cut. I am getting used to how the three different ways to define the start and finish, "Absolute, Incremental," and "Associative". Note the choices never finish the sentence, Absolute to **what**? I assume it is the tool plane, but maybe the Work Coordinate System, or maybe the construction plane.

A handy trick is you can copy and paste an operation with the same geometry, but doing a copy would crash the program. More issues with my old Win7 workstation. Instead, I just had to make the drill and tap operations from scratch. BobCAD CAM had no issues at all.

I could then change the tool, and set the parameters to peck drill or tap. One problem was the program changed the stock height to 1.4 inches. I had selected the top of the stock, with the "associative" radio button, but MasterCAM liked to put in 1.4, the distance from the tool plane to the top of the stock that got faced off previously. After figuring that out, I got the five holes drilled. That this took a half hour bothers me, but I did go down that automatic "rabbit hole, so I am sure I could do it faster. I am not sure I would remember all this if I was not using the program 8 hours a day every day, for 20 years.