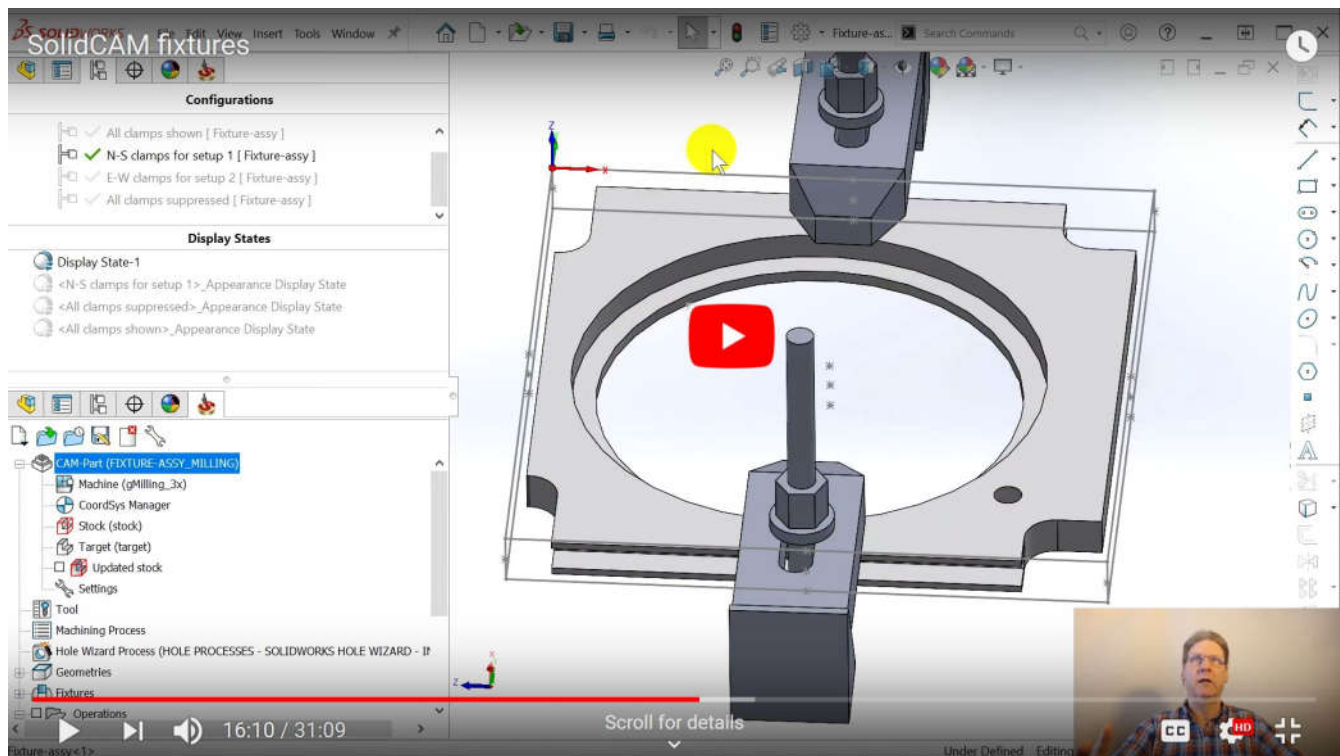




Rako Studios » Media » Suffering-with-software » SolidCAM fixtures

SolidCAM fixtures

It is important your CAM software allows for fixtures. Breaking a tool or ruining a spindle with a fixture crash is no fun.



Assembly zip file here.
Disjoint part file here.

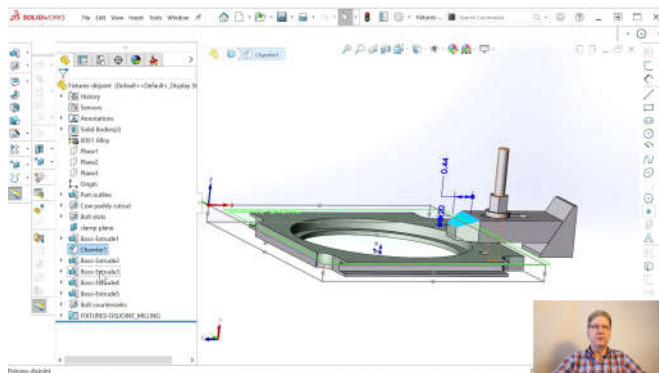
Before buying any CAM program, I wanted to look at how it handles fixtures. Every machinist I know has broken a tool by hitting a clamp or the vise. By modeling the clamps and vises in the CAM program, you can reduce the likelihood of a serious mistake on your CNC.

The experience of modeling some clamps made me glad I decided to limit my CAM package choices to those that work inside SolidWorks. There is a real comfort in staying with the tools and interface that I have used for almost 20 years. I was also pleased with how SolidCAM worked, though there were snags.

The SolidCAM tab tree has a line item called "Fixtures". Stupid me, that is where I thought I should define my clamps. Where you really have to put them is in the "Setup" line item. Double-click that, and there is a place to either list a fixture you defined in that "Fixtures" section, or create a fixture by pointing at the solids that make up your particular fixture.

One shortcoming of SolidCAM is that it cannot do fixture avoidance in a facing or contour operation, only in pockets. I would hope they add this capability in releases after 2020. I will note that SolidCAM did know how to avoid a fixture that was partially under the part, whereas VisualMill also avoided the piece of the fixture that was under the part.

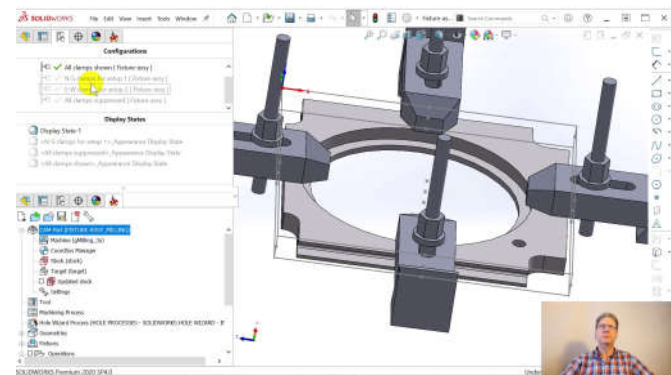
SolidCAM lets you define a fixture in a part file. If you can model the fixture as a disjoint body, then SolidCAM lets you select that disjoint part as the fixture. Be aware that when you first start to make the SolidCAM part, be sure that the "Target" of the dialog box is only pointing to the part you want to machine, and not the part as well as the fixture.



This screenshot shows a clamp built inside a part file. Note I made a "Clamp plane" a small distance above the part. If I built the clamp directly on the part, it would not be disjoint, and SolidCAM would just consider it to be part of the a thing you are trying to machine.

The fact that there is a "target" in that first dialog box when you start a new CAM part is what tripped me up for a few hours. The pocketing operation successfully avoided two of the clamps. Then, when I changed to a setup where those clamps were gone, the cleanup operation did one area properly, but skirted around the other clamp area.

I had remembered what a SolidCAM support person told me about a different file I sent him, that I had defined both the part and the fixture as target. Well, I didn't define them, SolidCAM saw them when I made the first Coordsys, and it selected both. I had never been checking for what the "target" was, I was just happy it got a green check-mark after I defined the Coordsys. By getting back to that dialog, and looking at the "target" I could un-select the fixture.



The right way to do fixtures is in an assembly. The screenshot above shows that same part now in an assembly, with four clamps added. Note that I also modeled the hold-down bolts, since you are even more likely to break a tool on those, though you might get lucky and just break the bolt before ruining your spindle.

Unlike all the other CAM programs I looked at, SolidCAM has an "external file" feature. What this does is copy your part or assembly file into a zip file with a PRZ suffix. If you rename it to a .zip, you can even look inside it.

The nice thing is that SolidCAM links the copy inside the .prz file to the original SolidWorks file. That's when, if you change the original file, when you open the PRZ file, it will warn you that the original part or assembly has changed and ask if you want to update it. If you update, SolidCAM shows you the operations affected and lets you supervise when you regenerate the toolpaths, to make sure things are OK.

I like this much better than SolidWorks CAM by CAMWorks, that will just do another auto-recognition of features and then make all the same erroneous operations it did the first time I tired auto-recognition. I like the external file feature since I want to keep my part files as small as possible. Note you could also make a single-part assembly file to hold toolpaths in SolidCAM or other CAM programs for SolidWorks, as long as they open an assembly file. I checked and VisualMill will do this.