



Author: Gloria Ngan (ngang@bxscience.edu)

Updated: 4/24/17

Sending Files

You might notice that CAD programs will give you many file formats to save the part as. Most machine shops use a CNC machine to create the parts. However, machining a part involves several steps:

1. Part modeled in CAD
2. CAM (computer aided manufacturing) program is used to convert CAD to G-code
3. G-code controls CNC machine to make part (our machine takes .NC files).

Types of Files

Fusion360 has a built-in CAM program, so worrying about file importing isn't necessary. To convert a CAD file into CAM, simply switch the workspace.

The best file format (if your vendor can read it) is the native format for the CAD system that you are using. If that doesn't work the next best file is a kernel file. Kernel files are generated by the kernel, or geometry engine that the modeler is based on. Kernel file types include:

- ACIS (from Inventor / Autodesk™)
- Parasolids (from Solidworks™, Unigraphics™, SolidEdge™)

If none of those work then you should consider a neutral file format. These are files that follow a standard set up by an industry. They are (in order of preference):

- STEP
- IGES surface file (wire frame is an option, but it doesn't help you in this application)

Almost all CAM packages will import STEP, or IGES, files. IGES files support "flavors", which means that you may have to set up options to get the transfer to work properly.

When you send out CAD files, make sure to rename them with a revision number. Let's say the original file name is "PART123.sldprt". You should rename the file to represent the revision #. An example is "PART123_Rev03.sldprt".

Surface Models

Surface models are made up of infinitely thin membranes that define the boundaries of the part. A surface model has no thickness. A solid model has thickness and material inside the boundaries, at least as far as the computer is concerned. Generally surfaces are used to create complex geometry that isn't easily created in solids.

Programs such as Rhino are often used to create surface models. If a surface model is not "water tight" (i.e. all surface and vertexes merge as expected) then the CAM package may not be able to import the surface model. A good approach here is to turn your surface model into a solid model and then send it to the machine shop.

As a result, surface models like .STL files are used for 3D printing, but not generally for CNC.

File Formats to Avoid At All Costs

Don't send a DXF or DWG file. Technically they work... but only works for simple parts. Both formats are two-dimensional. (That should work fine for plate with holes in it, as long as the machinist knows the thickness.) The other problem is that neither format supports surfaces or solids. They are wire frame only.

Because information is missing from these file formats, it is significantly more work (and opportunity for error) to develop CNC programs than working from the solid model. Other formats that aren't appropriate for sending to your shop are STL (stereolithography), VRML virtual reality markup language), and 3D DXF/DWG. All three formats are commonly used in other applications, but are not appropriate as a replacement for a solid model.

Round Tripping

Don't forget to "round trip" your part before sending it out. "Round tripping" a part is where take you exported file (Parasolids, STEP, IGES, etc.) and import it back into your CAD system. If the CAD system can rebuild the file into a solid model then the file is probably ready-to-send out. If your CAD system can't rebuild the file into a solid model, then you should try exporting it in another format and "round tripping" it again. If your CAD system can't rebuild the file into a solid model then it is likely that your local machine shop's CAM system will have similar problems. This step can save time by avoiding file format back-and-forth between you and the machine shop.

Drawings

Most CAD programs allow you to convert a model into a 2D drawing. PDF files are commonly used for drawings. They have several advantages including compact file size. They are also designed to be consistent in look and feel across a variety of computers and operating systems.

Acrobat files, and eDrawings, work great and have few issues. DXF and DWG-format files are more cumbersome, but will also work.

You can submit a file to your local machine shop without a drawing. However, creating a simple drawing with a minimal amount of information is a good way to help ensure that you get the finished part that you are expecting. Include:

- company title block (with a note about the drawing units [inches / SAE or mm / metric])
- part # or drawing reference number
- material callout
- material color (where applicable, an example is black vs. natural color Delrin™)
- isometric view of the part
- two dimensions for reference

Other things to include:

- Holes (tapped or reamed)
 - If the holes in your parts are loose, clearance holes, then it is not necessary to have them on the drawing. Holes that are tapped, reamed, or have a tight clearance should be included on the drawing.
- Slots (with tight tolerances)
 - Only slots with a tight tolerances need to be included on the drawing. Critical Tolerances
- All critical tolerances should be included on the drawing. This lets the machine shop know that they should be careful attention to those features.
- Surface Finish Callout
 - If your application requires a surface finish better than 128 micro inches, then you should include a surface finish callout.
- Finish (anodize, chromate, paint, etc.)
 - Clear anodize, clear chromate, yellow/gold chromate are readily available finishes for aluminum. Avoid colored anodize finishes because of long waits for the finisher to run that specific color again. Most plastics, such as Delrin™ and nylon, don't require a finish.