How to Prepare Your Solid Model for Rapid Machining

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Solid modelers have allowed us, as mechanical engineers and industrial designers, to design better products, quicker. Toolmakers now create complex tools directly from solid models of the end product. Clients can view visualizations of products before the first prototype is made.

One thing that has not changed however, is how most machine shops receive design information. The standard line is "send me a drawing and I'll get you a quote." Solid models are very precise. Solid modeling programs allows us as engineers, and industrial designers, to take design intent and turn it into a design that can be communicated to others. Why should your local machine shop not be able to accept your solid model and make you a part? What can be that complicated?

Many local machine shops should be able take your solid model and give you back a functioning, accurate part. We, as engineers and designers, just need to make sure that we give them all the information that they need. They, as shop owners and operators, need to have the software and skills necessary to work with the digital file that you send.

Traditional Process to Source Machined Parts

The traditional path from concept to part looks like this

1) product concept is developed by the industrial designer who generates ideations and renderings
2) engineer works with the designer to understand the design intent of the concept
3) engineer translates the concept into a cad model(s)
4) engineer creates a fully descriptive drawing from the cad model
5) machine shop receives the drawing and generates a quote
6) purchasing accepts the quote and contracts with the machine shop to make the parts
7) machine shop creates the machined parts
8) quality control at the machine shop checks the parts against the drawing (first article inspection) and ships the parts
9) parts are received by the company and passed along to the engineer who uses them to build the prototype and confirm the design integrity

There are a lot of steps for everyone involved. Usually the biggest delays in the process (other than designing the part, or assembly) are creating the fully descriptive drawing (Step 4) and the machine shop creating the parts (Step 7).

These steps work as part of a proven process to make accurate parts that reflect the design intent. However the process can be fairly long, especially when you consider that the part is recreated twice in the process (Steps 4 and 7). Solid modeling tools help streamline Step 4 (creating a part drawing), but this step is time consuming. Making a part from a drawing (Step 7) allows for interpretation and procedural errors.

The New Paradigm

Most machine shops use CNC equipment to make many of their customer's parts. These shops use software to generate the g-code that runs their machinery. Modern CAM (computer-aided-manufacturing software) looks and works a lot like the current cad software. Most CAM software will also read in models as wire frame, surfaces, or as solid models. Some CAM software will even import native CAD files from the more popular CAD packages.

Traditionally only a few engineers have pushed their local machine shop to accept CAD files. Most engineers have been content with the traditional process and it's long lead times. However there has been a growing pressure to shorten product development times. The first step was to use CAD. Then we started using rapid prototyping processes such as stereolithography. Stereolithography parts are not appropriate for many applications, especially where you want to build a functional, metal part.

Now there is a growing pressure on local machine shops to work from solid models. Let's face it, if you still design with a 2D CAD system, then you are in the minority. All the pieces are in place to take the next step towards higher productivity. The new process looks something like this:

1) product concept is developed by the industrial designer who generates ideations, renderings, and initial CAD models
2) engineer uses the initial CAD models to develop production CAD models
3) engineer creates a partially descriptive drawing from the cad model (where needed)
4) engineer transmits the CAD model and the abridged drawing to the machine shop via the web
5) machine shop receives the CAD model via the web and generates a quote
6) engineer gives the quote to purchasing
7) purchasing (or the engineer with credit card in hand) orders the part over the phone, fax, or via email
8) machine shop creates the machined parts directly from the solid model, referencing the drawing only when it is necessary to get information that wasn't provided in the model
9) quality control, at the machine shop, performs a cursory check of the part, verifying any critical features specified in the abridged drawing
10) parts are received by the company and passed along to the original engineer who uses them to build the prototype and confirm the design integrity

The end result is higher quality parts, as opposed to working only from a 2D drawing.

File Formats

The best file format (if your vendor can read it) is the native format for the CAD system that you are using. CAM package can often ready files from packages like SolidWorks™, or Pro/Engineer™.
If that doesn't work the next best file is a kernel file. Kernal files are generated by the kernel, or geometry engine that the modeler is based on. Kernal 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.

Once you get a format that works, stick with it for that vendor. When you send out CAD files, make sure to rename them with a revision number. Let's say the original file name is "PART123.STP". You should rename the file to represent the revision #. An example is "PART123_Rev03.STP". This helps the vendor to track which digital file represents which revision. Then you can specify "PART123_REV03.STP" on the purchase order. Specifying the revision makes it completely clear which version of the part the vendor should make.

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.

Why Surface Models Can Be Difficult for the Machine Shop to Work With

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.

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This is not to say that you should not use surfaces in creating geometry that you plan to send to your local machine shop. A good approach here is to turn your surface model into a solid model and then send it to the machine shop. If your CAD tool has problems creating a contiguous solid from the surfaces, then you have the opportunity to correct the problem before sending out the file. This will help save you time by avoiding submitting a file to the machine shop that isn't usable.

File Formats To Avoid

Every so often I hear some one ask why you can't just send a DXF, or a DWG-format, file to the machine shop. The simple answer is that you can, but it only works for simple parts. For the most part 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. Many parts have been programmed (for CNC) from DXF and DWG-format files. 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.

What Tolerances to Expect

Modern CNC machining processes are very precise. Tolerances vary from material-to-material, but you can generally expect to receive parts with the following tolerances (all dimensions are in inches):

Feature	**Typical Tolerance**
Machined Feature (body of the part)	+/-.002
Feature-to-Feature (both machined)	+/-.002
Drilled Hole	
Diameter (diameter less than 3/8)	+/- .003
Location (diameter less than 3/8)	+/-.002
Diameter (diameter greater than 3/8)	+/- .005
Location (diameter greater than 3/8)	+/-.002
Reamed Hole	
Diameter	+/-.0005

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Location	+/-.001
Tapped Hole	
Location	+/-.005
Slot	
Radius	+/- .001
Length	+/-.005
Flatness	
Simple Part (i.e. plate with holes in it)	.015
Complex Part (significant material removal)	.030

Simplified Drawing Basics

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 insure that you get the finished part that you are expecting. I would encourage you to, at a minimum, create a drawing with the following information and items:

- 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

Drawing Items

Other items you may need to include on the drawing are:

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.) - If you want a part quickly, then you should either not require a finish, or use an easily available finish. 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.

Profile Tolerance - It is a common practice to call out a profile tolerance for the entire part. However, if you do this you should keep several things in mind. Do not exceed the standard tolerances mentioned above. If a profile tolerance is greater than standard CNC process tolerances (see chart above), then you may unnecessarily add cost and a delay in getting your parts. If you include a profile tolerance, then you should include primary, secondary, and tertiary datums. Another issue to consider is that checking an overall profile tolerance economically usually requires a coordinate measuring machine. Generally, you only need to call out an overall profile tolerance when you are concerned about the overall quality of the part. If that is case, then you should consider using a different vendor to make your parts.

Flatness (if the flatness required is greater than .030) - It is important to keep in mind that flatness is a function of the flatness of the blank (before machining). Flatness can also change dramatically when significant amounts of material are removed from the blank. This change is flatness is caused by the removal of cast-in stresses. If you have a requirement for flatness greater than .030 inches per inch, you should probably show that on the abridged drawing. Depending on your application, you may want to use profile of any surface instead of a flatness callout.

Formats for Simplified Drawings

Adobe Acrobat™ 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. The Adobe Acrobat viewer is free. CAD systems such as SolidWorks™ can create Acrobat files directly without any sort of conversion.

eDrawings™ are another common format for transmitting drawing files. eDrawings™ is the brainchild of the folks at SolidWorks™. Originally, the eDrawings™ format was designed to give SolidWorks users a way to send out drawings that could be viewed on any computer. The eDrawings™ format also supports viewing (and rotating, measuring, etc.) parts, and assemblies, in three dimensions. There are eDrawings™ modules for many of the common CAD packages. You can find out more from http://www.edrawingsviewer.com/. The eDrawings™ viewer is free.

DXF and DWG-format drawings have several disadvantages. One is that they generally end up being viewed in the CAM software (after they have been imported). This is less convenient for the machine shop when compared to the extremely easy viewing of an Acrobat™ file. There is also the possibility for translation errors, especially with text dimensions that can make the DXF and DWG-format difficult to read.

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

Things You Should Assume Your Machine Shop Will Automatically Do

A good machine shop will deburr / break all edges and lightly sand all parts before releasing them to you. This step is often accomplished by lightly sanding the part with Scotchbrite™.

Quality vendors will also alert an engineer to possible issues with the part. However, this is not a substitute for design reviews and due diligence by the engineer, or designer. This type of feedback is incredibly valuable to us, as engineers and designers.

About the Author

Montie Roland is the President of Montie Design. Montie Design provides concept-to-marketplace product commercialization and product development services. Working as partners to our clients, we integrate mechanical engineering and industrial design to bring about products that balance performance, aesthetics, and cost. We can also make your prototype in our machine shop. The firm is based in the Research Triangle region of North Carolina, the Southeast's innovation hub. Call today at 1-800-722-7987 and let us bring out the innovator in you!

Montie enjoys finding innovative solutions to customer requirements. He has 15 years of experience engineering products in diverse market spaces including industrial, commercial and military. Montie earned a BS in Mechanical Engineering from North Carolina State University. He is also the President Emeritus of the Carolinas chapter of the Product Design Management Association (www.pdma.org/carolinas) and a founder of the RTP Product Development Guild (www.rtpproductguild.com).

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