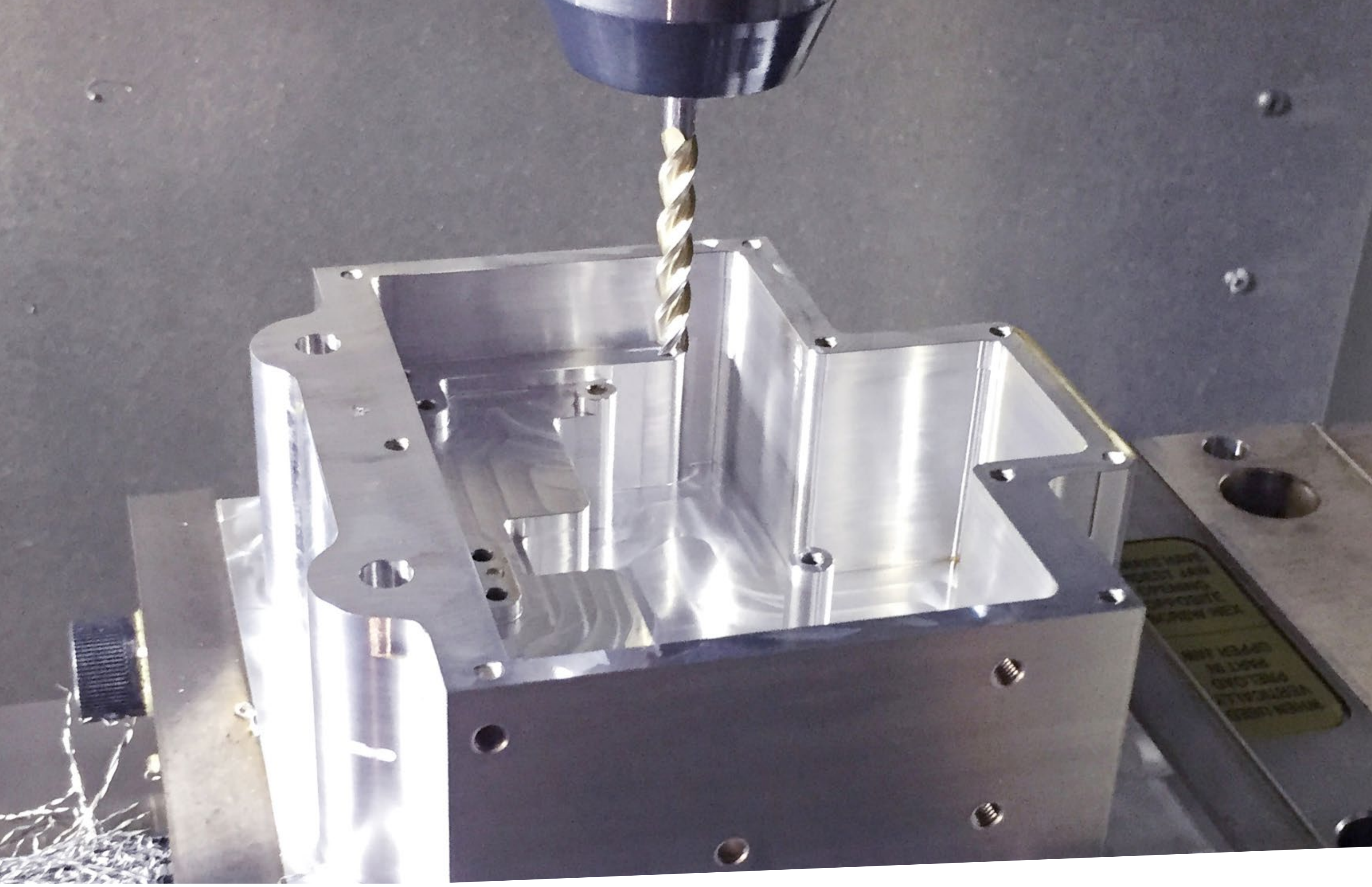
Cost Effective Design Tips for Machining Complex Parts



Introduction

Design for manufacturing requires cross-departmental collaboration to achieve high quality, cost-effective parts. In large companies with segmented departments, the engineering team typically draws up a blueprint and hands it to the purchasing department, which then begins the quote procurement process. Purchasing, often governed by the dollar, weighs its options, selects a manufacturer, and negotiates a contract.

When machining complex parts, the purchasing team often pushes the manufacturer to cut costs after the initial production phase is complete. By that point in the process, however, it is usually too late to rework a drawing for cost-efficiency, especially when the manufacturer was not the original designer.

Involving manufacturers at the front end of the design process provides engineers with the support they need to make small adjustments, subsequently avoiding these conflicts altogether. Purchasing departments, often focused on the larger, more expensive parts, may fail to see that smaller changes, such as opening up tolerances on several supporting components, will yield much greater aggregate cost savings.

“20% of parts probably have the ability to have 80% costs savings,” said Mike Sterling, Director of Operations at Ardel Engineering. “You just have to pick the right 20%.”

In this eBook, we offer design tips, suggestions, and considerations that will help you find that “right 20%.” From the ideation phase to drawing, machining, and inspection, devoting the necessary time to making mindful choices throughout the design process has a significant impact on overall manufacturing costs and production time.

Material Selection

Calling out the materials in your drawing is very design-dependent, but it's one of earliest points in the process at which your choices impact the bigger picture. Three main factors of design to manufacturing complex machine parts are:

1. Price **2.** Delivery **3.** Quantity

The first two considerations — price and delivery — are converted into metrics that can be tracked by buyers and members of the purchasing team because, as we know, time is money. Different materials typically have significant differences in cost and lead time.

The third factor — quality — is something that is expected in every job. If a design allows flexibility that won't compromise quality, different material options should be discussed with manufacturer ahead of time. By opening up a dialogue, the machinist can help to determine what alternative materials would drive down cost while maintaining the structural integrity of the design.

A common example would be calling out 304 stainless steel for a machined part. Because it is a frequently used alloy in the sheet metal industry, design engineers might specify a part with 304SS when they really mean simply any "stainless steel." In this case, 303SS is actually a much more machinable material. Having the flexibility to make this switch up front will maintain the part's desired material characteristics while saving both time and money.

Optimizing Performance and Reducing Cost

In addition to a material's machinability rating, other important considerations to optimize performance include strength and durability. Annealed, heat treatable alloys without any explicit heat treatment specification, for example, will add needless cost to the design. There are also times when plastic would be a more machinable substitute for metal, depending on the requirements of the application.

Another major material consideration should be geometry. The length of time required to machine a part, which is largely guided by geometric complexity, is one of the biggest overall cost drivers. The shorter a part is, the faster it can feed through a machine and the lower its production cost. Conversely, the larger a tool is that can be used on a given corner, the faster it will be able to process the material.

Designing a part to match the size of the raw material is possible when dealing with low tolerance and cosmetic requirements. Tight tolerances, however, can quickly drive up cost — especially when they're difficult to machine and measure. Therefore, a better understanding of the machining process can allow the engineer to specify the appropriate tolerances that best serve the project's specific needs without creating cost overruns.

Material specifications, part geometry, and functional or cosmetic requirements should be discussed between the engineer and the manufacturer prior to production. The machinist cannot independently make the decision to substitute a material on a new design; however, active collaboration between the two teams helps to determine the most quality and cost-effective material or cosmetic selections.

The Drawing and Print Phase

Since machinists tend to think in visuals, developing an accurate and detailed drawing is a crucial step in the design process. In a way, the print functions as a contract; the manufacturer is agreeing to create something that looks like what is represented in the design blueprint, so it's of the utmost importance that the specifications show exactly what you want before sending the design to the purchasing team.

To ensure that you've created a clear, robust blueprint for the manufacturer, we recommend following the checklist outlined below:

- **Use names and part numbers** — Make sure to clearly label and identify each part in the design. Use the unique part number and be as specific as possible.

- **Specify materials** — As previously mentioned, part materials should always be specified, as such considerations can have serious ramifications on cost, cycle time, and lead time.

- **Specify surface finishes** — Sometimes engineers call out a surface finish range, such as a 16-32 surface finish, for example. Requiring the machinist to hit inside of a range in this way can be difficult, especially because modern tooling often makes surfaces much smoother than the requested minimum. Although there are times when creating texture is necessary, it's best to discuss the part's function with the machinist ahead of time to achieve the intended surface.

- **Tell the machinist what a part is for and explain how it will be used** — While this can be challenging in industries where non-disclosure agreements are prevalent, providing as much detail as possible regarding the function of the piece helps manufacturers with customization and making critical decisions about material changes, tolerances, and other factors that could help to reduce costs.

- **Provide a completed print to the best of your ability** — From an estimating standpoint, it's imperative to have a completed blueprint with tolerancing, surface requirements, edge break requirements, and more considerations. An accurate CAD model makes it much easier to program CNC mill, lathe, and Swiss screw machines, in addition to inspection equipment such as CMMs (coordinate measuring machines). However, some design situations simply do not allow for such models. For example, engineers working on unique projects using models from 1960s military and aerospace designs are only provided with a 2D blueprint. In these cases, manufacturers often end up drawing the CAD model, which equates to more time and money.

- **Don't fake a dimension** — Modern CAD systems sometimes allow engineers to call out specifications that can't actually be implemented in the working environment. The program will allow you to drill a ¼" deep hole with a ¼" tap as well, or make a note saying two holes are 2" apart, when the design really shows them to be 1½" apart. These numbers may complete a drawing in the short term, but will not work when manufacturing begins. Once machines are programmed with the CAD created model or design, they will make what is actually represented in the model — not what the drawing may inaccurately indicate.

- **Pay close attention to tolerancing** — Be mindful of what you're requesting, even if it looks good on a screen. When putting rounds on the corners of a box, for example, an engineer might call out .060" radius on every edge, even the outsides. This would require a special rounding tool that adds complexity, cost, and time. Instead of limiting the manufacturer by mandating strict requirements, the preferred method would be to draw the outside features and then provide a range, such as "Break all edges .005-.015 of an inch.

- **Dimension consistently from the same edges** — Always lay out locational feature dimensions from three main datum planes or edges to avoid tolerance stacking. Especially when you're looking at a design from many different views – front, top, left, etc. – it is critical to maintain a consistent datum structure throughout the drawing so that all dimensions originate from the same three planes when possible.

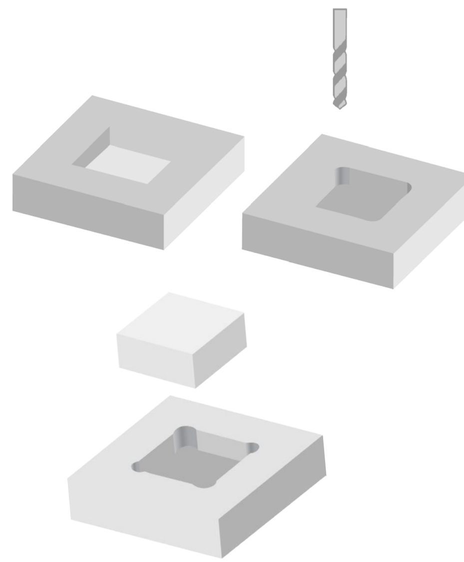
Although these extra steps may sound tedious, cross-checking your blueprints with this checklist and discussing them with your manufacturer during the design phase are only small upfront investments compared to the time and money required to fix mistakes after a part has been made. Accounting for these considerations will save your company from having to make adjustments much further down the pipeline of the design for manufacturing process.

The Machining Phase

When manufacturing complex parts, engineers can make many considerations to limit possible complications during the tooling and machining parts of the process.

First, it is important to account for **endmill (EM) radii** and design with standard EM sizes in mind. This means avoiding sharp, square, inside corners in a design. For example, if a design will tolerate a ¼" radius and the manufacturer uses a ½" endmill, the endmill will actually pause and chatter when it turns a sharp corner, resulting in a poor surface finish.

Instead, the preferred solution would be to call out a .275 inch radius in the corners, so that when the manufacturer uses a ½" endmill it can actually drive all the way around the corner instead of coming to a complete stop before making a hard 90° turn.



For internal radii, if you would like to use the largest end mill possible, try to avoid using a nominal inch-sized radius. Although fractional EMs are the most prevalent in the United States, it is possible to buy metric tools to achieve a broader variety of radii.

As mentioned earlier when discussing design and blueprint creating, **thread tapping** can get increasingly difficult and expensive depending on hole depth. When specifying a drilled and tapped hole, allow the drill portion to go much deeper than the required depth of the thread. If a hole is 1/4" deep, for example, the machinist will only be able to create good threads down to roughly .180 of an inch.

There are two considerations to take into account when tapping a feature: first, there's always a lead on the tap, and second, the chips will likely fall into the hole during the tapping process. If the machinist is using a vertical mill and there isn't any extra space in the hole, gravity naturally causes the chips to fall in and pack at the bottom of the hole. To avoid this buildup, always design holes that will accept the thread gauge and allow the drill to reach the full depth required for the thread length.

Finally, limit **machining risk factors**. As a general rule, it's always better to design assemblies with smaller, simpler parts than to try to build one large, complex part. When possible, design multiple parts that can fit together to diminish any high walls or other clearance issues that could potentially arise.

Proper Use of GD&T

Geometric Dimension and Tolerancing (GD&T) is a Means of dimensioning and tolerancing a drawing with respect to the actual function or relationship of part features which can be most economically produced. A specific language used on mechanical drawings to communicate geometric requirements of all necessary features for components and assemblies. This language includes design aspects such as Form, Profile, Orientation, Runout, Location.

Engineers use the GD&T methodology to define complex relationships between part features. When used correctly it ensures fitment and enables bonus tolerance that is not allowed using linear tolerances. Thereby making complex designs easier to manufacture while achieving functional requirements.

The proper use of GD&T is powerful and can make your parts more manufacturable. However, it is a vast topic with many different interpretations; engineers should carefully research GD&T before ever implementing it in the design process. The below reference chart can serve as a preliminary resource on this complex philosophy, but we highly encourage you to perform additional outside research.

GEOMETRIC TOLERANCING REFERENCE CHART

<table>
<tr><th>Type of Tolerance</th><th>Geometric Characteristic</th><th>Symbol</th><th>Can Be Applied to a Feature (Surface)</th><th>Can Be Applied to a Feature of Size</th><th>Can affect Virtual Condition</th><th>Datum reference used?</th><th>Can use M modifier</th><th>Can use S modifier</th><th>Can be affected by a bonus tolerance</th><th>Can be affected by a shift tolerance</th></tr>
<tr><td rowspan="4">Form</td><td>Straightness</td><td>⏤</td><td rowspan="7">Yes</td><td>Yes</td><td>Yes*</td><td rowspan="4">No</td><td>Yes*</td><td rowspan="7">No ●</td><td>Yes □</td><td rowspan="4">No</td></tr>
<tr><td>Flatness</td><td>⏥</td><td rowspan="3">No</td><td rowspan="3">No</td><td rowspan="3">No</td><td rowspan="3">No</td></tr>
<tr><td>Circularity</td><td>○</td></tr>
<tr><td>Cylindricity</td><td>⌭</td></tr>
<tr><td rowspan="3">Orientation</td><td>Perpendicularity</td><td>⊥</td><td rowspan="5">Yes</td><td rowspan="3">Yes*</td><td rowspan="5">Yes</td><td rowspan="3">Yes*</td><td rowspan="4">Yes □</td><td rowspan="4">Yes ○</td></tr>
<tr><td>Angularity</td><td>∠</td></tr>
<tr><td>Parallelism</td><td>∥</td></tr>
<tr><td rowspan="3">Location</td><td>Positional</td><td>⌖</td><td rowspan="2">No</td><td rowspan="2">Yes</td><td>Yes</td><td>Yes</td></tr>
<tr><td>Concentricity</td><td>◎</td><td>No</td><td>No ●</td><td>No</td><td>No</td></tr>
<tr><td>Symmetry</td><td>⌯</td><td>No △</td><td>Yes △</td><td>Yes △</td><td>Yes △</td><td>No △</td><td>No △</td><td>No △</td><td>No △</td></tr>
<tr><td rowspan="2">Runout</td><td>Circular Runout</td><td>↗</td><td rowspan="4">Yes</td><td rowspan="2">Yes</td><td rowspan="2">Yes</td><td rowspan="2">Yes*</td><td rowspan="4">No</td><td rowspan="4">No ●</td><td rowspan="4">No</td><td rowspan="2">No</td></tr>
<tr><td>Total Runout</td><td>⌰</td></tr>
<tr><td rowspan="2">Profile</td><td>Profile of a line</td><td>⌒</td><td rowspan="2">No</td><td rowspan="2">No</td><td rowspan="2">Yes**</td><td rowspan="2">Yes ○</td></tr>
<tr><td>Profile of a surface</td><td>⌓</td></tr>
</table>

Notes

* When applied to a feature-of-size
** Can also be used as a form control without a datum reference
○ When a datum feature-of-size is referenced wih the MMC modifier

Per ASME Y14.5 M-1982

□ When an MMC modifier is used
● Automatic per rule #3
△ The symmetry symbol's characteristics were not included in the version of the chart that this chart is derived from. The symmetry symbol was dropped from the Y14.5M standard around 1982 and re-added around 1994.



Symbol	Modifier	Notes
Ⓕ	Free state	Applies only when part is otherwise restrained
Ⓛ	Least material condition (LMC)	Useful to maintain minimum wall thickness
Ⓜ	Maximum material condition (MMC)	Provides bonus tolerance only for a feature of size
Ⓟ	Projected tolerance zone	Useful on threaded holes for long studs
Ⓢ	Regardless of feature size (RFS)	Not part of the 1994 version. See para. A5, bullet 3. Also para. D3. Also, Figure 3-8.
Ⓣ	Tangent plane	Useful for interfaces where form is not required
Ⓤ	Unilateral	Appears in the 2009 version of the standard, and refers to unequal profile distribution.

The Takeaway on Design for Manufacturing

The biggest theme pulsing through these content sections is the importance of involving manufacturers at the front end of the design process; if possible, communicate with the Sales Engineer or machinist from the beginning, especially when developing new products. From a manufacturer's standpoint, if teams haven't had any communication up until post-prototype production, it's often too late to find a more cost-effective solution.

After prototyping, testing, and fixture building, a single feature change on an individual component could affect a multitude of other components within the same design. A small change that could have been simple to implement at the beginning ends up requiring numerous engineering change orders and revised prints. Furthermore, the design for manufacturing (DFM) activity when preformed late in the process could force the team to perform a lot of guesswork about the original design, increasing confusion, the potential for error or a complete redesign.

In critical applications like aerospace, in which every component must be flight-tested, post-production changes can have devastating ramifications. By that point in the process, neither the design nor the process can be altered without possibly having to flight-test the part again. All of this, of course, adds on time and money. Proactive discussion about cost-effective production is the key to avoiding these impasses.

The headform particle counter component on Ardel's website is an example of a design for manufacturing problem and solution developed through effective collaboration between designer and manufacturer. The part's original design was comprised of four separate pieces (a block and three tubes) pressed together with a very tight positional callout.

The Ardel team knew that it would be faster, cheaper, and more accurate to streamline the process and make the whole component out of one single piece. The CNC machinists, designers, and development engineers all sat down together before production to revise the design, build a quality product, and avoid post-production pushback. That's how design for manufacturing should be.

About Ardel

Ardel Engineering & Manufacturing has specialized in precision CNC machining of metal, plastic, and engineering grade material components for 50 years. We work closely with engineers to aid in the design process and build their products in the best way possible.

The Ardel facility hosts an extensive list of state-of-the-art equipment and tooling in our facility, including high quality inspection and vision equipment. After production, we also provide a host of value-added services including anodization, grinding, honing, passivation, and more.

We serve every industry from aerospace and medical, to commercial, automotive, and more. Customers are invited to visit our sample parts gallery to view examples from our diverse body of work. By focusing on proper documentation, precision machining quality, and excellent customer service, we achieve quality, delivery, reliability, efficiency, confidentiality, and partnership with every individual customer.

We hope you will find these design tips helpful on your next project. For more information about how we can work together to create the right part for your application, please contact Ardel today.

View our
PROJECT GALLERY