



# Introduction to Design for Manufacturing

An Online Continuing Education Course for Engineers

**Course Number: I-4001**

**Credit: 4 Hours / 4 PDH / 4 CPD**

# Introduction to Design for Manufacturing

Charles A. Seifert, P.E.

## Chapter 1 Introduction

Many newly designed products fail. This can happen for a number of reasons, such as having missing features, being impossible to build, having too high a cost, or not making it to market in a timely fashion. This course will focus on design and ease of manufacturing—factors that significantly impact the aforementioned causes of product failure.

The substance of the course is basic design principles, which can be applied across a multitude of manufacturing processes. The course will provide “rules of thumb” to follow when using a given manufacturing process. The major processes to be discussed are injection molding, die-casting, extrusions, machining, and sheet metal work. One key term is “Design for Assembly” (DFA), which includes reviewing how component parts will be put together in manufacture of a specific product. This course will consider DFA and go over, in detail, a set of rules for designing products with assembly in mind. Then, it will review material selection for each process and follow it with a consideration of the importance of, and techniques for, performing a tolerance stack up on the assembly. Lastly, this course will cover Design for Disassembly (DFD) and how to design for the environment.

### Essential Definitions

- Asymmetrical – Having unequal dimensions (or features) on either side of a central axis.
- CNC – A term used in machining when the process is automated; “Computer Numerical Control”
- Draft Angle (“draft” or “taper”) – A small angle added to the side of a part to allow it to easily release from a mold. For example, this might lead the bottom of a rib to be thicker than the top of a rib.
- Design for Assembly – A set of guidelines for parts when they are used as part of an assembly.
- Design for Manufacturing – A set of guidelines to follow to improve any design when preparing it to be manufactured.

- Field Failure – When a product breaks or stops working by the end-user (customer).
- FMEA (Failure Mode and Effects Analysis) – A process to discuss product (potential) failures and how they may affect the design.
- “Form, fit, or function” – A phrase used in design to discuss whether a feature in a part is there for aesthetic, practical, or task-specific reasons. Form-related (aesthetic) features influence appearance (or another sensory impact of the design). Fit-related features impact how one component combines with another. Function-related features directly affect whether a manufactured product will perform the task(s) for which it was designed.
- Fixture – A mechanism that is used to hold a part while other operations are being performed.
- OD – Outer Diameter (ID is Inner Diameter)
- Robustness – A term used when describing a part and whether it is designed to meet all of its mechanical expectations.
- Shelf Life – The time from when a product is made until the time when it will no longer perform to the TDS specification. Many adhesives have a one year shelf life; after that time period an adhesive might not cure to the strength needed.
- Symmetrical – When all of the features and/or dimensions on one side of the part are identical to those on the other side. Symmetry is usually assessed along an “axis” (e.g., a line that denotes one side of the part from the other side).
- TDS – Material Technical Data Sheet – Information about the material, how it performs, and how it should be processed.
- Tempering/Tempered: “hardened” (such as when a process acts on aluminum to harden it)
- Tolerance Stack up – The process of determining if parts will fit together without having any interference.

## Chapter 2

### Basic Design Principles

Designing a part can seem simple but quickly become complicated. An experienced engineer should think about previous designs and what could have been done differently to make each one a little better. Some engineers have a difficult time releasing designs into production. They might spend design time analyzing each feature and hypothesizing about how—by testing and redesigning—they can save production costs, never quite knowing when a product design is good enough. In this chapter, we'll go over very basic design rules for mechanical designs. The principles set forth here might help an engineer to discern when his/her design is ready for production and when additional testing and/or redesign might be prudent.

#### **Basic Guidelines for Design:**

The basic principles for design can be summarized in eight simple rules.

**Symmetry** – Design each part with symmetry in mind. Either design the part so that the assembler does not care which way it is oriented or so that the part is so obviously not symmetrical (i.e., is asymmetrical) that the assembler knows exactly in which direction to orient components during assembly.

**Minimize part count** – Injection molding and die cast parts allow a design engineer to reduce the number of parts that will be used. For example, replace screws with snap-fit parts. The more features that can be combined into one part, the more likely it is that recycling can occur. It is a designer's goal to lower assembly costs for the product being designed. Eliminating unique parts and reducing the number of parts can help. Using fewer component parts can significantly reduce the assembly time. Reducing the number of component parts also reduces the different types of failures—thereby increasing overall reliability.

**Use standard parts** – Using “off the shelf” parts not only saves on tooling and special assembly tools but can lead to significant cost savings for the product. Make certain that the “off the shelf” component parts will not be discontinued by the manufacturer, or a redesign will need to occur. Do not forget that one's own company may have parts that are common to other designs, and by using those parts, one can save in unneeded tooling. Using parts that are already in stock reduces the need for new suppliers to be added and new inventory numbers to be managed; the hidden cost savings can be huge.

**Minimize special features** – Reduce tight tolerances or very smooth surface finishes, each of which leads to higher manufacturing costs. Do not design a tolerance that the manufacturing process cannot hold. And stay away from design features that cannot be produced using standard fixtures or tooling.

**Avoid difficult to assemble parts** – All parts must easily fit together. The amount of time it takes to assemble all of the parts can easily “out-weigh” the benefits of the new design’s unique feature(s). Always look for ways to make the assembly process “error-proof”. Multiple, sequential assemblies of the same part might bring fatigue, and with fatigue comes an increase in human error. Easy assembly helps to cut down on fatigue-associated assembly mistakes and waste.

**Avoid extra (or unnecessary) features** – If the feature cannot be made with normal processes, try to remove it. As an example, do not design holes that are perpendicular to the way the mold opens and closes. This design feature would either require side pulls that increase mold costs, or it would add a secondary operation (like additional machining) to the process.

**Ease of fixturing** – Design parts with features that will allow the use of a fixture (or fixtures) during the assembly process. Using fixtures to hold parts while other processes are being performed is essential. Using two human hands and/or dexterous (grip) force during assembly will increase the difficulty of the assembly task. A designer should assume that one or more fixtures will be needed and should design features into each part in order to hold it/them in place—thereby reducing the difficulty of assembly.

**Consider automation** – Design parts that can be assembled by automation, as well as by humans. Choose parts that can be assembled with power tools, rather than by hand. Design parts that will automatically feed into an assembly feeder in the correct direction every time.

Each of the items listed above will be discussed in more detail in the remainder of this course.

## Chapter 3

### Manufacturing Processes

The design guidelines given here are general. Note that each vendor/manufacture has unique capabilities, and the designer should know what they are when sending a part out for quotation. The guidelines given here are aimed at minimizing redesigns. It is easy to find reasons to stretch a design above and beyond the given guidelines (for example, in order to minimize costs and optimize profits). When this happens it is important to share those needs with the vendors one uses.

#### **Designing for Injection Molding**

A summary of some of the basic design rules for injection molding is as follows:

- Similar and/or uniform wall thickness
- No sharp corners, except possibly at mold parting lines
- Generous drafts
- Clear path for material flow
- Symmetry, whenever possible
- Allowing a location for a gate

Design uniform wall thicknesses. The reason for an injection mold to have unvarying wall thickness is so that—when the part is cooling, after being ejected from the mold—the part does not warp. For those designs where the part may warp anyway, sometimes extending the cooling time in the mold or placing it directly into a cooling fixture may be necessary. Warp risk increases with long, thin parts. Another reason to have consistent wall thickness is to allow the material to flow more easily throughout the entire mold cavity. If this not possible, the tool designer can make accommodations to the mold design, but that may be costly and hold up production. So, note that wall thickness plays a major part in design of plastic parts.

Add ribs, because ribs in plastic parts can be very valuable. Ribs make a part more rigid without adding much weight. The best way to determine the added rigidity is to calculate the cross-sectional moment of inertia. A simple example is as follows.

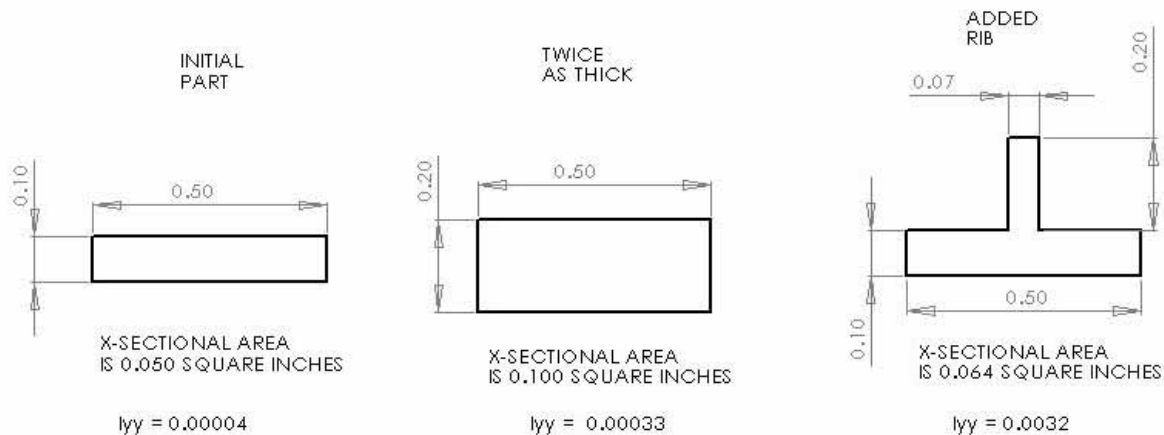


Figure 3.1

The design in Figure 3.1 starts with a simple part that has a thickness of 0.10" and a width of 0.50". Keeping the width the same throughout this exercise, one can see that by doubling the thickness (middle view), the stiffness increases in the Y (vertical) direction. The moment of inertia goes from 0.00004 in<sup>4</sup> to 0.00033 in<sup>4</sup>. One gains 7.5 times the stiffness by simply doubling the weight. But, a designer can do even better by adding just one, simple rib as shown in the third view. If a designer were to add a 0.20" X 0.07" rib, the weight would increase by only 2.00 times and the moment of inertia increases to 0.0032 in<sup>4</sup>. That is an 80 times greater increase in stiffness. When it is possible, add ribs to a design in the Y direction.

One of the problems with adding ribs is shrinkage where the rib is located. If a rib is added to a wall where it is added, the wall thickness should be less than three times the thickness of the designed rib can reduce the shrinkage.

Beware when designing ribs. If a rib is designed poorly, the wall thickness will be even the best design. Refer to Figure 3.2 for some examples of poor rib thickness, but embed ribs in the wall. A uniform wall thickness in boss design is not the best. The point of failure might be at the pieces (small ribs) or at the side wall (of the case).

*To view the remainder of the course material and to take the quiz for PDH credit, you must purchase the course.*  
*Close this window and click "Add to cart" on the product page.*

shrinkage where the rib is located. If a rib is added to a wall where it is added, the wall thickness should be less than three times the thickness of the designed rib can reduce the shrinkage.

can be difficult. If a rib is designed poorly, the wall thickness will be even the best design. Refer to Figure 3.2 for some examples of poor rib thickness, but embed ribs in the wall. A uniform wall thickness in boss design is not the best. The point of failure might be at the pieces (small ribs) or at the side wall (of the case).