Design for Manufacturability: CNC Machining

Design for manufacturability (DFM) is the process of engineering and designing products and components in an optimal way that will:

- Ease all manufacturing functions (fabrication, assembly, shipping, etc.)
- Ensure cost reduction, without compromising on quality, compliance, safety, or customer satisfaction

DFM is a proactive way of addressing potential problems in the design stage. Early consideration of manufacturing issues reduces the product development time and cost and reduces the time to market.

Materials and Material Forms
The strength versus machinability rating needs to be considered in deciding which material is the best choice for each component.

For example, an annealed material can have the same properties as a heat treated and tempered material, which is more difficult and expensive to machine. Aluminum can have better performance and machinability than some grades of steel. Soft metals such as aluminum, brass and magnesium are easily machined. Steel, titanium and exotic alloys are stronger, more dense and harder, thus much more difficult to machine.

When choosing materials, it is important to allow the use of different forms, such as bar stock and plate. Bar stocks are usually half the price of cut plates (at least in aluminum). Being limited to only one option can significantly affect the cost, as well as the lead time.

Geometry and Cutting Tools
The length of time it takes to machine a part is one of the biggest cost drivers. The cutting tool's rigidity and strength is most often the determining factor in how much time it will take to machine a certain component. The cutting tool is selected based on design.

The shorter the tool (relative to its diameter), the faster it can feed, and the lower the cost will be. As the length of the cutting tool increases, the feed rate is cut down significantly (in order to avoid deflection and breakage) and the cost goes up. Generally speaking, a good length to diameter ratio is less than 3:1. Each time the length doubles, the feed rate is cut in half, and then some. 8:1 is the upper limit and the most expensive to cut. Considering these facts early in the design process and adapting the geometry of the part accordingly can bring significant cost benefits.

Tolerances
Tight tolerances can require complicated fixturing, long machine set-up, longer loading times and a higher scrap rate, leading to more expensive components. It is recommended to use the loosest tolerance that still provides a robust design and serves the function of the component. In some cases, a part's tolerances can be made loose enough to allow the use of laser or waterjet cutting, reducing the cost significantly.

It is common for engineers to rely on the tolerance block to help communicate their needs such as .xx=.01”, .xxx=005”, .xxxx=.0005”. While this is easy to do, it may incur additional costs.

For example, if a .005” tolerance is too loose, it doesn't mean that the next logical step is to jump to a .001” tolerance. Using an intermediate tolerance (such as .0025”) may be much easier and less expensive. Tolerances must be intelligently applied to each individual feature. When working with tight tolerances, a small change can make the difference between staying on budget and going over.

Consulting with an experienced manufacturer in the early developmental stages can help ensure an effective, intelligent design. Since 80% of a product's cost is determined during this step, embracing design for manufacturability techniques could result in spectacular cost reductions.