DESIGNING A **COST-EFFECTIVE** CNC MACHINED PART

chap2@chap2.com | 920.648.8125

TABLE OF CONTENTS

INTRODUCTION

Chapter 2 Incorporated has operated as a trusted contract manufacturer for decades, providing comprehensive CNC machining services at competitive prices. Over the years, we've learned how to keep costs down without jeopardizing the quality of our finished parts. Through our strong supply chain network and carefully selected strategic partnerships, we can accommodate orders across industries and volumes, treating each project to a fully customized approach from design to testing. While CNC machining lies at the core of our company, we have expanded our in-house contract manufacturing services to include painting, welding, metal fabrication, assembly, and custom machine building. We also have a full-service tool room to meet the needs of our clients for custom workholding fixtures, jigs, dies, fixtures, and molds, we have you covered.

You'll be surprised what a big change small modifications can make when it comes to your bottom line. This e-book on designing a cost-effective CNC machined part addresses ways to save money, avoid expensive mistakes, and make the right decisions, starting with designs and materials.

At Chapter 2, we proudly serve the following industries:

- Lawn and Garden
- Automotive
- Military
- Agriculture
- Medical Equipment
- Construction
- Food Packaging
- Renewable Energy

As a full-service contract manufacturer, we always look forward to welcoming new companies and industries into our client family.

PART DESIGN – DESIGN FOR MANUFACTURABILITY

1. AMOUNT OF ROTATIONS/ANGLES

Limiting the number of angles and rotations can keep costs down by reducing tooling and fixture changes. With 5-axis CNC machining, which has two additional rotational axes, you can eliminate the need for different machine setups and cut complex geometries. However, it's the more expensive route to go and often avoidable with simpler designs.

2. FEWER OPERATIONS/SETUPS

LESS FIXTURING

In a perfect world, all parts would require machining on just one side. In reality, most parts are more complex and require machining on multiple sides. When designing parts, consider the machine and tools used to produce them. With single side pieces, cheaper machines can produce the parts quickly. Single side pieces typically require just one fixturing, which can save time and reduce error rates.

For parts with multiple sides, design the part so that the manufacturing process requires only one clamping. Additionally, when possible, design features on the same axes to limit rotation. Machining angles along various axes increases the amount of fixturing and may require more complex equipment —both of which increase the per unit cost.

LESS ROTATIONS WHILE MACHINING

When you create a metal part that you can produce with a 3-axis rather than a 5-axis manufacturing process, it's certainly faster and cheaper to do so. When you need to design parts with multiple sides, put as many features on the same axis as possible to avoid more complex machines and multiple fixtures.

3. EFFECTS OF TYPE OF MACHINES USED

You can choose a 3-axis or 5-axis machine, a specific type of lathe, and determine whether HMC or VMC would work better for a particular part. Let's take a look at each of these factors from the standpoint of efficiency, cost and productivity.

VERTICAL MACHINING CENTER (VMC) VS HORIZONTAL MACHINING CENTER (HMC)

In general, a horizontal machining center or HMC is cheaper than a vertical machining center or VMC. However, you have to compare that to the greater productivity of an HMC, which is at least twice as productive.

An HMC delivers higher spindle on times due to less movement and part repositioning. For example, using a VMC to make a rectangular part requiring machining on six sides requires six position adjustments. Meanwhile, the operator only has to change the position three times on an HMC.

Surveys published by machining magazines and universities indicate that HMCs average 85% spindle on time while VMC's average 25% spindle on times. So, in an 8-hour shift, the HMC cuts for about 6.8 hours, while a VMC cuts for about two hours.

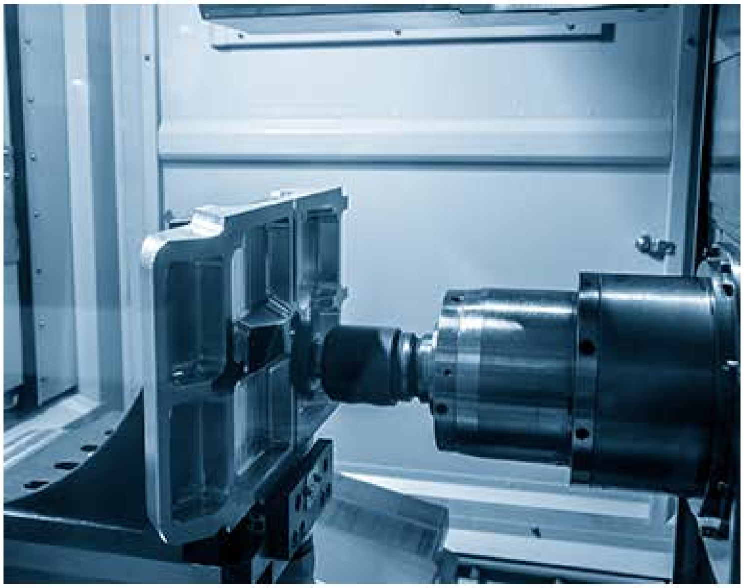
In short, HMCs improve productivity, reduce labor costs and take up less floor space due to their greater productivity. Despite the additional upfront outlay, switching to HMCs can reduce your capital cost and prevent the need to move to a larger facility.

VERTICAL MACHINING CENTERS

VMCs machine on a single access. However, you can add trunnions and indexers to machine on multiple axes. These machines are not as robust as HMCs and require more adjustments by the operator. Basically, VMC part designs with multiple sides require a unique set up and fixture for each side. This adds time and requires additional inspection to ensure that each part meets tolerance standards. It typically adds additional fixturing as well, which can also slow down the process. Many machinists like VMC's because they are easy to program and set up.

HORIZONTAL MACHINING CENTERS

Horizontal machining centers machine on multi-dimensional axes with a single clamping. These machines are easy to set up, fixture and program. By requiring a single clamp, HMCs eliminate the need for multiple fixtures, creating more accurate parts, reducing waste, and decreasing labor costs. Additionally, HMCs improve part quality, accuracy and tolerance compliance.



LIVE TOOL LATHES/Y AXIS LIVE TOOL LATHES/SUB SPINDLE LATHES

Definitions that may be helpful for this section include:

- **Turret:** The turret holds the tool and indexes it accordingly. While the spindle holds the part in place, the turret positions or moves the tool as required. Slides allow the part program or operator to move the tool in multiple axes simultaneously.
- **Spindle:** A CNC spindle describes the shaft in the middle of the rotational axis of a machine tool. Depending on the reference, it may also include bearings and attachments. Spindles rotate around an axis guided by a CNC controller.
- **Trunnion:** A pivot which allows you to rotate or tilt the part. It is used in 5-axis machining.

- Standard CNC lathes can turn parts in two-axis machining. The part spins and a rigid mounted tool peels the material off. Additionally, the tool moves in two axes.
- Live tool lathes allow you to control the spindle, which forms the C axis. The turret, which holds the tools, can spin a tool within the tool area. This allows you to drill off-center holes. During operation, the spindle stops and the live tool drills holes while the main spindle remains stationary. This type of machine can also use an end mill in the live tool area and mill the part being held in the spindle. This can be done with two-axis movement and also with the main spindle (C axis) rotating during milling. With a live tool lathe, the spindle rotation can work with the two machine axes.
  Y axis live tool lathes combine the capability of regular live tool lathes. However, they have an additional angle position along the X axis. This allows for fast, precise motion when drilling and milling with the live tools.
- Sub spindle lathes hand off parts on the main spindle to a parallel sub spindle. This enables machining on the side or back of the part.

5-AXIS MACHINES

A 5-axis machining center can handle 4-axis machining, but it also has a dual trunnion that allows for the part to be rotated. All five axes can move simultaneously, allowing you to machine features at axes other than straight down from the spindle.

Benefits of 5-axis machining include:

- Increase capacity and productivity
- More efficient production (more on spindle time)
- Operator flexibility and reduced labor cost

However, 5-axis machining centers are more expensive than standard CNC machining centers.

4. STANDARDIZATION OF FEATURES

Standardizing of features cuts down on the number of tools needed to machine parts. For standard tools, for example, the ratio between the cutting and shaft diameters is 2:1, which gives you the maximum cutting depth. Some machine shops customize undercut tools to accommodate different depths. However, this adds to the time and cost per part.

FEWER TOOL CHANGES REDUCE MACHINING TIME

Each tool change takes time. So, designing parts with similar hole sizes, including inside and outside radii, can reduce the number of tool changes and tools needed to complete the part. For example, consider a simple part with six sides and eight holes. If all eight edges require a different radius or chamfer (symmetrical sloping edge), you may need eight tool changes which can significantly slow down productivity. Conversely, if each edge requires the same radius or chamfer, it only requires one tool setup.

The same concept applies to holes. We recommend using standard drill bit sizes but any diameter over 1 millimeter is possible. You can machine holes using a drill bit or end mill tool. Which one you pick depends largely on the size. Machine shops often use standard drill bits and then finish the holes with boring or reaming tools to achieve tight tolerances.

The recommended maximum depth is four times the nominal diameter. However, many parts require 10 times the normal diameter and some machinists go much deeper. Using an end mill tool can help you achieve non-standard diameters.

If possible, design dowels, tapped holes and clearance holes with common diameters to reduce the number of tools required. A tapped or threaded hole contains threads added to the clearance hole. Some metal parts require this when they can't use nuts and bolts.

To what extent can a tap drill be used as a dowel prep hole? Tap drills can drill precisely threaded holes, and many shops use taps—resembling a screw with teeth — to create internal threads in predrilled clearance holes. Others use a thread mill that appropriately interpolates the threads. Using this method, you can use a single tool to create thread sizes with standard pitches.

FEWER TOOLS AT SETUP: HOW TO REDUCE SETUP/CHANGEOVER TIME

Using one of the examples above, if you require eight tool changes that each take 15 minutes, you spend a significant part of the shift changing tools. Compare that to a single tool change of 15 minutes total. Now multiply that by every operator and every part you produce. It's easy enough to see why reducing setup time can save you time and money.

LESS INSPECTION REQUIREMENTS

You have to inspect each feature requiring a different tool. If a part uses eight different radii that would mean checking each radius verses checking just one. If you use different hole sizes, you may need a plug gauge for each thread size. Plug gauges, or pin gauges, gauge the inside of holes of manufactured parts and assemblies. Whenever possible, use simple parts with similar features to reduce the amount of gauging needed.

STANDARD STOCK TOOLS VERSUS CUSTOM TOOLS

Design parts you can make with tools already in your shop to avoid higher production costs. Designers should consult with machinists to use off-the-shelf tooling whenever possible. This means using common tap sizes, radii and hole sizes for parts.

Of course, not all parts will have standardized features. However, you can use this best practice to help your customers save money on each part.

5. INTEGRATE INTERNAL RADII

You can save time and cost by avoiding 90-degree angles and using integrated internal radii. Cylindrical CNC milling tools naturally leave behind a curve instead of right angles. To get around this, you can include internal radii in the part design instead of relying on expensive end milling processes to achieve right angles. Keep in mind that using large internal radii allows for larger milling tools during production, resulting in faster machining times per unit.

6. AVOID DEEP HOLES AND THREADS

We've already touched upon this briefly. However, it's worthwhile to emphasize this as a design tip. By avoiding deep cavities, you can save machining time in the cutaway process. That’s because you may need special tooling for cavities deeper than four times the diameter of your tool. This type of specialization will increase production costs, eating into your profits or increasing costs.

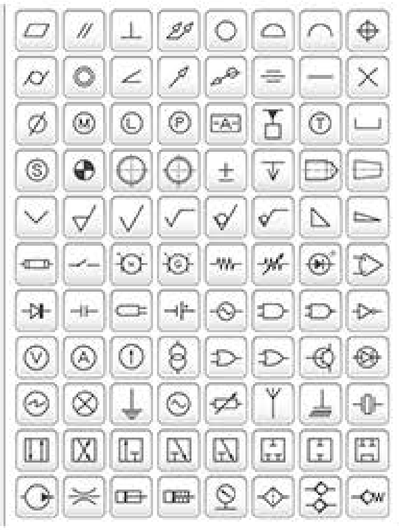
Follow the same precaution on thread depth. Deeper threads often require nonstandard tools, driving up your costs and lowering productivity.

7. AVOID THIN WALLS

CNC machining allows for tight tolerances and can produce thin walls of 1 mm thickness. Machining thin walls requires slower speeds to avoid damage due to vibrations. Also, producing thin walls requires repeated low cutting depths. This adds a lot of time to the process. To get around this problem, use thicker walls in your part design whenever possible, greatly increasing throughput.

DIMENSIONING AND TOLERANCING

1. USE OF GD&T



Geometric Dimensioning and Tolerancing describes design intent and helps manufacturers control variations and tolerances during part production. Moreover, GD&T represents a philosophy that incorporates the impact on manufacturing in the design process.

GD&T offers numerous advantages:

- Saving Money. GD&T results in more accurate designs by indicating tolerance ranges for optimal production. By communicating maximum tolerances, GD&T often allows manufacturers to further cut the cost per unit.
- Ensuring Tolerance Requirements. A thorough GD&T better guarantees that the finished parts meet the dimensional and tolerance requirements for the part specifications.
- Facilitating Digital Design Methods. Clear GD&T data adapts well to 2D/3D CAD files.
- Maximizing Uniformity. GD&T symbols provide a consistent language that takes the guesswork out of the manufacturing process.

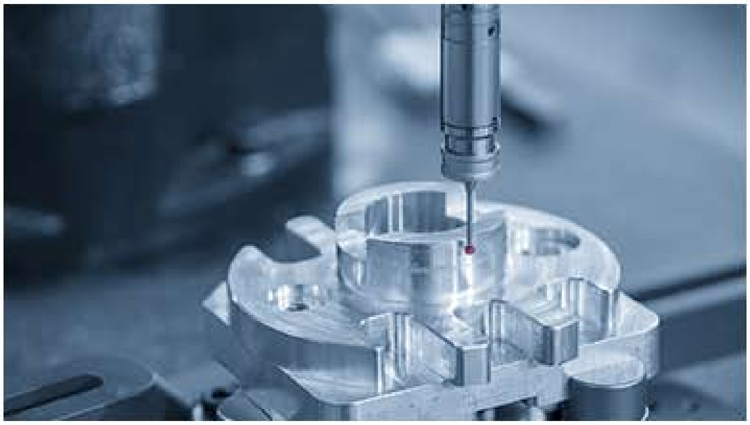
DRILL AND TAP

Think of the GD&T as one-third of what is specified in each axis. So, if you use .010” true position, the tolerance is .0033” in each direction for the hole. To do this, all tapped holes would need to be thread milled. The typical walking or moving of a drilled hole is .003”-.005”. If you do need a tight true position, allow for a large enough hole diameter tolerance and specify the use of maximum material allowance. This will help reduce time needed to reach unjustified tolerances.

DATUM STRUCTURE SETUP

Use a large surface to specify your plane and major datums (machine offsets). Features such as holes are critical to the function of the part but may not serve well as a coordinate measuring machine (CMM) setting feature. Instead, use large surfaces to align the part. Errors can occur when using small features to project over a large surface. You can use tight location rather than datum to identify significant holes.

2. GENERAL TOLERANCING



In general, do not make tolerances too tight or over-engineer parts without factoring in manufacturability.

DIAMETERS

The tolerance on parts must hold for each part manufactured. Therefore, it’s crucial to get the tolerance correct during the design phase. This can prevent placing stringent restrictions that don't take the manufacturing process into account. Also, this may be seen as a sign of complacency and can cause major cost overruns if done repeatedly.

OUTSIDE SURFACES AND SHAPES

Some features do not directly impact the functionality of the part. In this case, use wide open tolerances to give manufacturers the opportunity to optimize the production process.

USE STOCK MATERIAL SIZES, IF POSSIBLE

When designing, use stock sizes whenever possible. This can minimize machining and result in a lower cost per part.

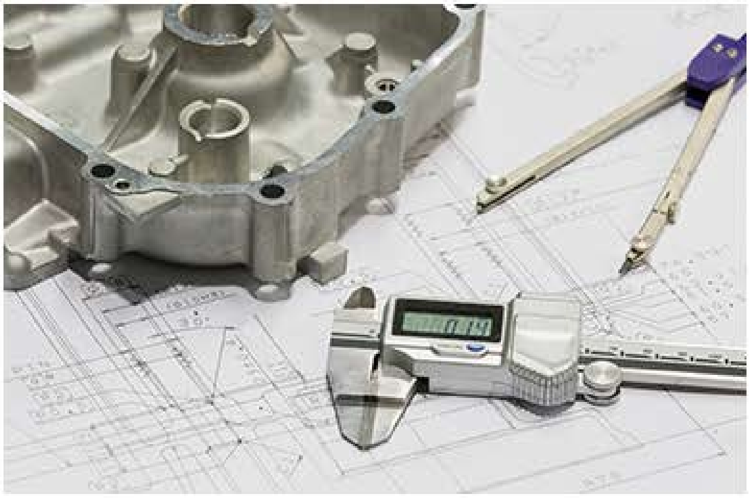
SURFACE FINISHES

Minimize surface finishes to save on machining time and wear and tear on the tools. This includes avoiding lettering or text engraving when possible.

3. BE CLEAR AND CONCISE WITH YOUR DRAWING

Detail is important but don't over-tolerance to make the part more difficult to machine.

Drawing clarity ensures accuracy and completeness. Draw dimensions and tolerances outside of the part drawing and communicate the function of the part. When you design for the loosest possible tolerances, you can reduce cost and give more flexibility to the manufacturing team. All part design should include general tolerances located at the bottom of the specifications. Of course, tighter tolerances inside the drawing will supersede more lenient tolerances.
Do not describe part production within the design document. This limits the flexibility for the manufacturing team and may cause additional tooling and fixture costs.



WORK HOLDING AND TOOLING

1. INCLUDE CLAMPING FEATURES

Think about how the part is going to be held through the entire manufacturing process. If the part you design needs more than one clamping position, that significantly impacts fixture adjustments and inspections required.

2. TOOLING STANDARDIZATION

There are three considerations when it comes to designing for standardized tooling, including the following:

- Reduce the number of tool changes
- Reduce time to set up tools in the machine
- Design with intent to use standard tools versus custom tools

3. TOOL ACCESS

Design with tool access in mind. Also, the type of machine used to manufacture the part plays a role in the design (3-, 4- or 5-axis, live tooling HCM vs VCM).

MATERIALS AND SECONDARY SERVICES

1. RAW MATERIALS

Consider a more machine friendly (softer) and less expensive material such as 1018 rather than 4140. If corrosion resistance and high strength isn't critical, specify 316L or 304 rather than 17-4 PH stainless steel.

BILLET, CASTINGS OR TUBING

Of the three methods, casting is typically most cost-effective, but may not work for parts requiring tight tolerances and extreme durability. You can use tubing to improve accuracy and productivity on previously handmade parts.

- **Casting:** Aluminum is the typical material for casting because it is a light metal with a wide range of applications. For example, cast aluminum parts are typically used in manufacturing engine blocks, pistons and cylinders for the automotive industry. Other materials may include brass, iron or steel.
- **Billet:** Parts made with billets of aluminum or other materials are typically stronger than cast parts. That's because parts created from solid billet machined down on a mill result in elegant, precise parts. You can often find billet upgrades on the aftermarket.
- **Tubing:** Tubing is manufactured through an extrusion process where the tube is drawn from solid billet and extruded into a hollow form. The billets are first heated and then formed into oblong circular molds that are hollowed in a piercing mill.

FINISHING REQUIREMENTS

1. POST PLATING SURFACE FINISH REQUIREMENTS

Sometimes, it's not possible to avoid post plating during the surface finish process. However, post plating costs more than plating the part before machining. Post plating guarantees the entire part receives the necessary coating or finish material. Therefore, it's important to research cost efficiencies in the type of coating and other factors.

2. PLUGGING

It's important to use high quality plugs to make the molds for small and large components. Typically, plugs are used for machining plastic and composite materials.

3. MASKING

Masking (stop-off application) is used to isolate portions of metal parts during the finishing process. Masking protects the untreated parts and helps reduce waste.

OTHER IDEAS FOR COST REDUCTIONS

1. VOLUME AND RUN QUANTITIES

The more parts you buy, the lower the cost.

- Amortize setup over larger quantity of parts
- Amortize NRE (non-recurring engineering) costs over larger quantity of parts

2. HAVE A 3D MODEL AVAILABLE

Printing a 3D model of the part allows your customers and machinist to note any discrepancies and issues prior to the test run. This can also reduce NRE and programming time.

TAKEAWAYS

When it comes to designing cost-effective CNC machined parts, it's important to consider every aspect of the manufacturing process. From loose tolerances to choosing the appropriate finishing method, your efforts to reduce waste and keep costs down, begin with the very first 3D drawing you produce.

Feel free to browse our photo gallery to get an idea of the CNC milling, machine turning and assemblies we can produce for your company. We also offer custom paint and tool and die services to our partners.

Contact Chapter 2 Incorporated today with your questions or to get started on your cost-efficient parts design.

www.chap2.com

920.648.8125

REFERENCES

HMC VS VMC:

HMCs vs. VMCs: 3 Reasons to Consider a Horizontal Machining Center (southernfabsales.com)

Horizontal Machining Center Vs. Vertical Machining Center: Is a VMC or an HMC Better? (smarterfinanceusa.com)

"SURVEYS DONE BY AMERICAN MACHINIST AND THE UNIVERSITY OF MICHIGAN ..."

Okuma | Vertical vs. Horizontal Machining Centers

--These studies quotes in two of three articles above. Haven't found original studies.

STANDARDS:

How to design parts for CNC machining | Hubs

GDT ADVANTAGES:

Learn the Benefits of the Proper Use of GD&T (ardelengineering.com)