TECHNICAL GUIDE

CNC Machining

HIRSH PRECISION
PROTOTYPE PRODUCTION

INFORMATION@HPPI.COM | 303-530-3131

Introduction

This comprehensive guide covers the most critical aspects of CNC machining.

CNC (Computer Numerical Control) machining is a subtractive manufacturing process, where parts are made by removing material from a solid block (called the blank or the workpiece) using a variety of cutting tools.

This is a fundamentally different way of manufacturing compared to additive (i.e., 3D printing) or formative (i.e., injection molding) technologies. The material removal mechanisms have significant implications on the benefits, limitations and design restrictions of CNC.

Contents

INFORMATION@HPPI.COM | 303-530-3131

CNC Milling

CNC milling is a process where computer-guided tools remove material from a workpiece to form a specific shape or object.

In contrast to a lathe, the cutting tool in CNC milling rotates and moves around a stationary workpiece.

The manufacturing industry primarily uses milling to cut materials from a workpiece, creating a wide variety of items in different sizes and shapes. Milling machines are known for their high precision and tight tolerances.

Unlike turning, which works only with cylindrical parts, milling can create a wide range of shapes and contours in materials that are challenging or even impossible to machine with other methods.

CNC milling offers a number of key benefits, including:

Scalability
CNC milling is versatile for any production size, from custom projects to mass production. Its repeatability and speed make it ideal for prototyping and large-scale manufacturing, reducing costs as production volume increases.

Tolerances
CNC milling achieves exceptional precision, with tolerances as tight as 0.025 mm. This level of accuracy is essential for industries such as aerospace, automotive, and medical devices.

Variability
CNC milling can precisely create various features such as pockets, threads, chamfers, slots, and cavities, in addition to producing different shapes.

CNC MILLING

Anatomy of a CNC Mill

Spindle

The spindle holds the cutting tool in place

Control panel

This is where the computer interface is located and how the operator controls the machine

Column

The Column is the main frame and support of the machine; it holds other components in place

Saddle

The saddle is attached to the column of the machine; it supports the worktable

Worktable

The worktable is located on top of the saddle; it is where the operator places the workpiece and uses a workholding device

Base

The base is what provides support for the whole machine on the ground

CNC MILLING

Types of Milling Machines

The range of milling machines is vast, including types like planer, gantry, turret, and bed. Broadly, CNC milling machines are categorized into two main types: vertical machining centers (VMCs) and horizontal machining centers (HMCs).

In VMCs, the spindle stays fixed while the table moves beneath it. Sometimes, the table moves up to the spindle, or the spindle moves along the Z-axis. These machines are very sturdy, allowing for precise component production. However, they have a smaller work area. VMCs can operate with 3 axes, 4 axes, or 5 axes.

In HMCs, the spindle is positioned horizontally. These machines are perfect for long production runs because they can handle up to three times more work than a VMC, as long as there is enough work to keep them running.

However, HMCs are more costly than VMCs. They allow for continuous production since a block of material can be set up on the machine bed while another part is being made. The spindle can quickly switch to the next ready block, making changeovers fast.

3-Axis Mill

In 3-axis CNC milling machines, the cutting tool or workpiece can move in three directions: X-axis (left and right), Y-axis (forward and backward), and Z-axis (up and down). To access specific areas, the workpiece needs to be manually repositioned.

Pros

- Straightforward programming
- Efficient high-volume production
- More cost-effective than using a 5-axis mill

Cons

- Requires repositioning for more complex parts
- Often can't reach sections of a workpiece
- Not well-suited for parts with complex geometries

5-Axis Mill

5-axis mills have two more rotary axes than 3-axis ones, giving the cutting tool access to more areas through additional movements of the workpiece. 5-axis mills come in two types: indexed, where the tool pauses as the workpiece rotates, and continuous, where both rotate simultaneously without stopping.

Pros

- No need to reposition the workpiece
- High accuracy on parts with complex geometries

Cons

- More expensive than 3-axis mills
- Larger footprint and more expensive

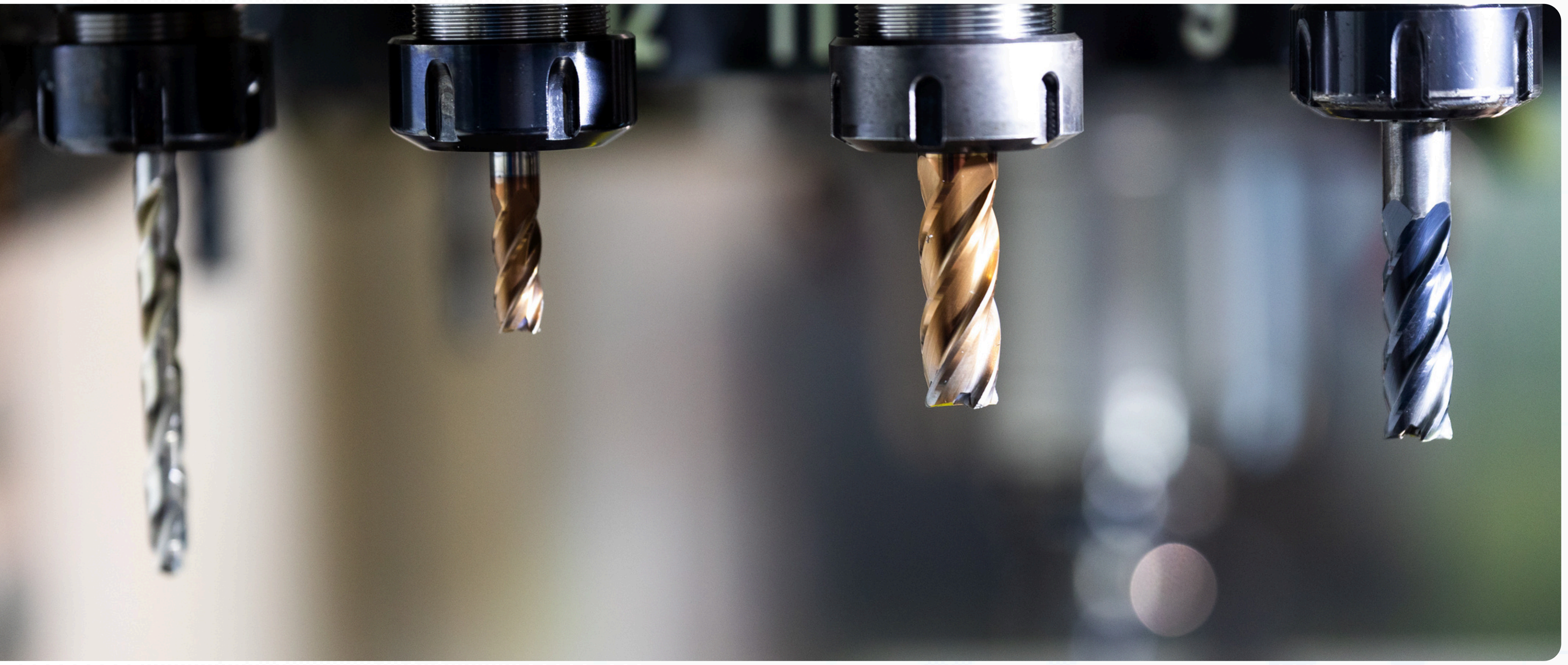
INFORMATION@HPPI.COM | 303-530-3131

CNC MILLING

Milling Tools

The tools handle all the cutting tasks. They are usually placed in a tool holder and inserted into the spindle as needed.

Various tools are required to create a complete part, as manufacturing does not follow a "one size fits all" method. Below are the most commonly used tools in a standard machining setup.



Related Read: How to Choose the Right End Mill

End Mill

An end mill is a versatile tool that cuts in three directions and comes in styles like flat, corner radius, roughing, ball, and taper. It is defined by its flutes, helix angles, base, and coating materials.

Face Mill

A face mill is crafted to cut large surface areas, known as facing. Its cutting edges are on the tool's perimeter, and the teeth are typically made of carbide inserts.

Thread Mill

A thread mill creates threads by rotating around the shank in a helical pattern to cut the thread shape.

Slotting Cutter

These cutters are designed to create t-slots along a part's length. Due to their shape, they must enter and exit from an open side of the material.

HIRSH PRECISION
PROTOTYPE PRODUCTION

INFORMATION@HPPI.COM | 303-530-3131

CNC MILLING

Milling Operations

While many milling operations are straightforward, it's beneficial to explore them in greater detail.

Face Milling

Face milling is highly valued in manufacturing for creating precise and smooth surfaces, even on large pieces. It can form pockets and steps in metal by flattening and smoothing the top. A face mill has cutting edges on its disk-like face, which shave off material horizontally from a workpiece until the preset depth is reached.

Slot Milling

Slot milling uses a slot cutter, similar to a circular saw blade, to create slots like grooves or trenches. This tool cuts into the sides of materials, while an end mill can also cut on its side and face. It essentially carves channels in the material, such as keyways and other grooves.

End Milling

The end milling tool has cutting edges on its end and sides, enabling it to create slots, pockets, and contours. This versatile tool is popular because it can cut straight down like a drill bit and also move sideways.

Thread Milling

Thread milling creates the spiral threads on screws and bolts. This tool, with its multiple cutting edges, can make threads in different materials and sizes, even for complex designs. It rotates around the material, following the thread path to cut the threads. While its main job is to create threads, it performs this task exceptionally well.

INFORMATION@HPPI.COM | 303-530-3131

CNC MILLING

Milling Operations

While many milling operations are straightforward, it's beneficial to explore them in greater detail.

Shoulder Milling

Shoulder milling uses tools like end mills to cut material from the side of a workpiece, forming flat, 90-degree angles similar to steps.

Side Milling

Side milling uses a cutter or end mill to shave off thin layers from the side of a workpiece, making it perfect for crafting flat or shaped surfaces.

Profile Milling

Profile milling is ideal for creating complex shapes and surfaces, using tools that have multiple cutting edges and a ball tip for smoothing.

Gear Milling

This milling op specifically makes gears and gear teeth, such as those found in car engines or clocks. It employs gear hobbing or milling cutters to shape the gear teeth by removing material. While simple gears can be crafted with manual machines, CNC machines are generally used for more intricate gears.

INFORMATION@HPPI.COM | 303-530-3131

CNC MILLING

Milling Operations

While many milling operations are straightforward, it's beneficial to explore them in greater detail.

Angle Milling

Angle milling shapes parts with beveled edges and angles. It uses special tools like cone-shaped or flat cutters that can be tilted to the needed angle. This tilt is achieved with a tilting arbor or a machine with four or more axes.

Form Milling

Form milling creates detailed and contoured shapes on surfaces using either specialized cutters that match the desired shape or standard cutting tools. This process is frequently employed in the automotive, aerospace, and mold-making industries, often utilizing CNC machines.

Straddle Milling

Straddle milling uses two milling cutters placed side by side on an arbor to create parallel slots, grooves, or surfaces on a material. This dual-cutter setup allows for faster completion, effectively cutting the time in half. It is commonly used for keyway milling and machining flat surfaces on opposite sides of a shaft.

Plain Milling

Plain milling may not have an exciting name, but it is an essential manufacturing technique. It involves using a flat, horizontal cutting tool to remove material from a surface, creating square or rectangular features.

Milling Ops

CNC Turning

CNC turning is a machining process where a lathe is used to rotate the metal while a cutting tool moves in a linear motion to remove metal along the diameter, creating a cylindrical shape. The cutting tool can be angled differently to create different forms.

Unlike working on a mill, the cutting tool is positioned against the workpiece, which is rotated by the spindle.

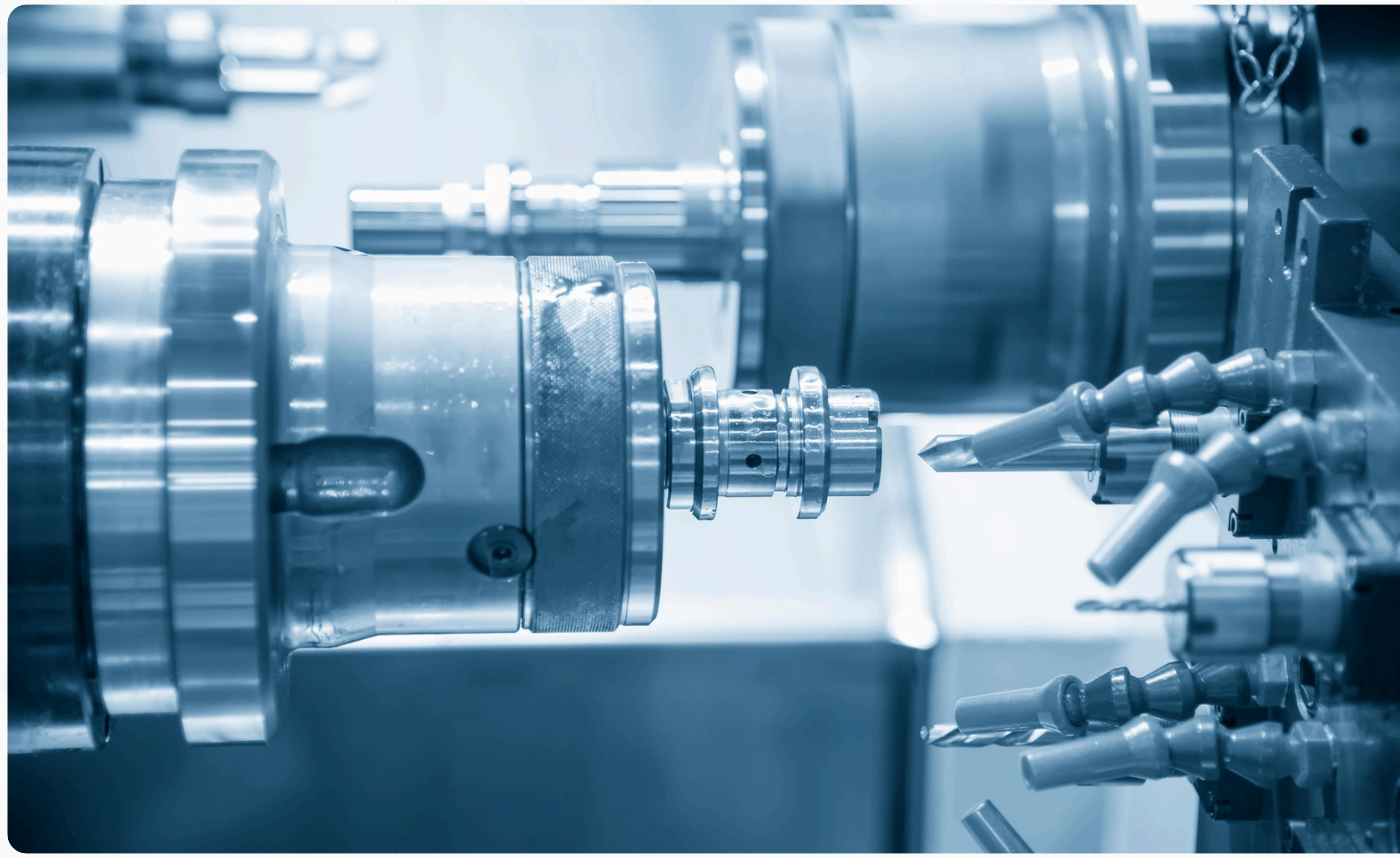
Turning produces rotational, typically axi-symmetric, parts with many features, such as holes, grooves, threads, tapers, various diameter steps, and even contoured surfaces.

CNC turning boasts several noteworthy advantages, including:

Cost and Volume
Lathes offer the lowest cost per part compared to all other CNC operations. They're also capable of very high production runs, making them an ideal machining process for manufacturers who want to reduce costs once they progress past the prototype phase.

Tolerances
Turning can create parts with extremely high tolerances. Because of the high tolerances and surface finishes that turning can offer, the process is often used to add precision rotational features to a part whose basic shape has already been formed through a different method.

Labor Requirements
Turning does not require a highly skilled operator of the machine. To handle a CNC lathe, a machinist can complete a set amount of coursework and earn certification from an accredited industrial training organization.



Due to the nature of CNC turning, only rotatable parts can be used on the machines. This can limit you as far as what types of components you can subject to turning or force you to rely on additional procedures and machines.

CNC TURNING

Anatomy of a CNC Lathe

Headstock
The lathe headstock powers the machine, holding and centering the workpiece.

Tailstock
The tailstock, positioned opposite the headstock, secures the workpiece's end.

Tailstock Quill
The quill is a cylindrical mount that holds the Morse taper and lets you attach chucks, drills, and more to the tailstock.

Tool Turret
This is the working end of a CNC lathe, where different tools are positioned for various operations.

Lathe Bed
This is the main structure that holds the machine together and supports the working parts.

Chuck
The chuck holds the workpiece in place at the center of the lathe, and while it connects to the headstock, it's a separate piece.

Foot Pedals
Foot switches, or pedals, let operators control machine setup or unloading while keeping their hands free.

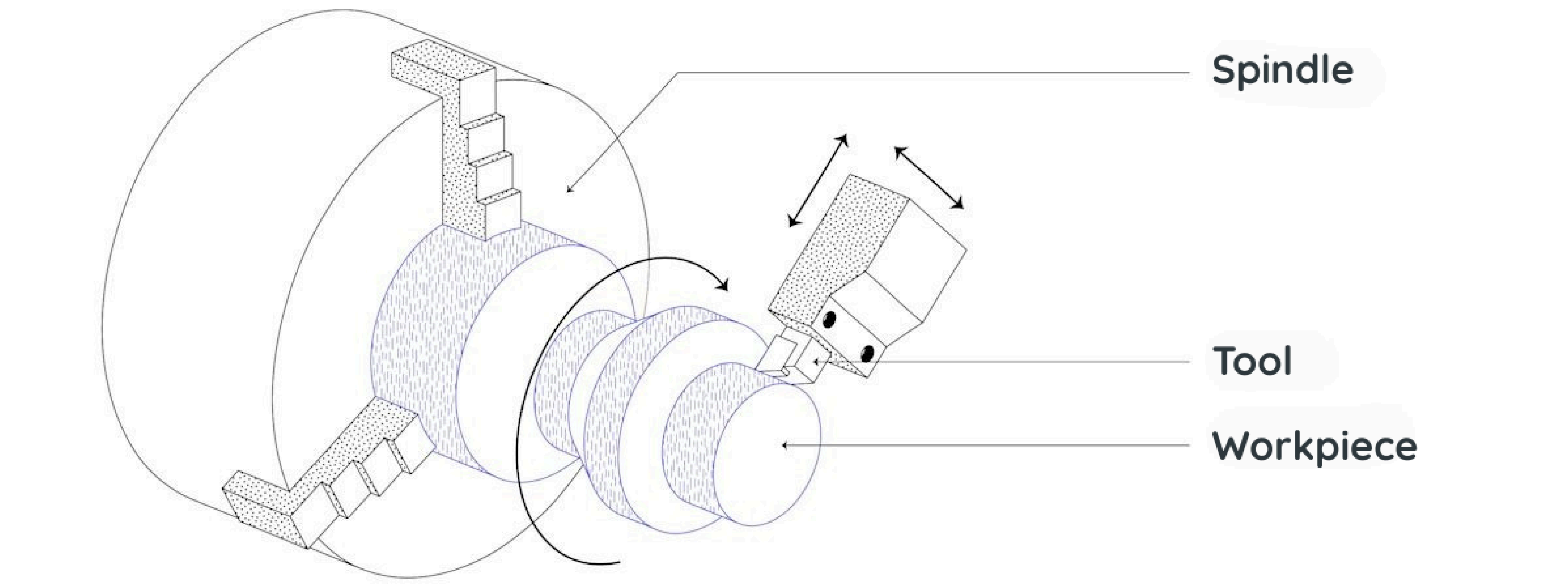
Control Panels
Lathes have two main control parts: the machine panel and the control panel. The former adjusts the tool's settings and cutter position and is used to make small manual adjustments to the machine's axes. The latter is for entering and editing programs.

CNC TURNING

Types of Turning Machines

CNC turning machines come in two main types: vertical and horizontal. Additionally, it's crucial to understand the distinction between a CNC lathe and a CNC turning center.

A CNC turning center can perform various operations, whereas a lathe is specifically for turning. A lathe cuts a rotating workpiece, and turning is the general term for this rotational cutting process.



Turning centers are versatile machines with multiple functions and axes. They can include a C-axis, Y-axis, and driven tools on the turret, allowing them to perform not only turning but also milling, drilling, and tapping operations.

A vertical turning center combines features of both a turning center and a vertical lathe. It is ideal for machining shorter components that do not need a sub-spindle, tailstock, or bottom turret, helping to minimize the machine's footprint.

INFORMATION@HPPI.COM | 303-530-3131

CNC TURNING

Types of Turning Machines

Horizontal Lathe

In CNC turning, a horizontal lathe is the most commonly used lathe and, as the name implies, features a spindle set horizontally.

A horizontal spindle allows the workpiece to be placed between the headstock and tailstock.

Horizontal lathes are adaptable and can manage various sizes and shapes of workpieces.

The horizontal setup simplifies loading and unloading workpieces and conducting different machining tasks.

Well suited for producing shafts, rods, tubes, and other cylindrical parts.

Vertical Lathe

As its name hints at, vertical lathes have a spindle that is set vertically.

This makes it easier to manage larger and heavier workpieces and manage chip removal more effectively since gravity helps clear the work area.

The workpiece is placed on a rotating table that can hold large and heavy items.

The design offers greater stability for machining large parts, minimizing deflection and vibration risks.

Well-suited for producing large-diameter components like wheels, gears, and large bearings.

Swiss Lathe

A Swiss lathe is a specialized lathe that's used for machining small, cylindrical components.

The workpiece moves along the Z-axis while the cutting tools remain stationary, improving precision and stability.

A guide bushing stabilizes the workpiece, enhancing accuracy by minimizing deflection and vibration.

Swiss lathes have several tool stations for quicker and simultaneous machining.

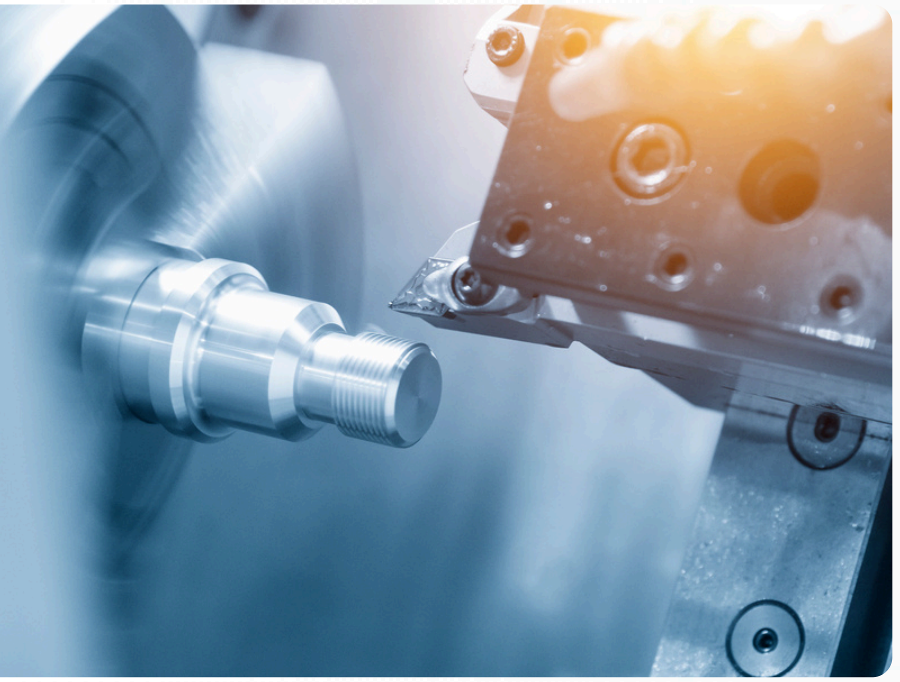
Well-suited for producing screws, pin, and other small components with complex geometries.

CNC TURNING

Turning Tools

The tools handle all the cutting tasks. They are usually placed in a tool holder and inserted into the spindle as needed.

Various tools are required to create a complete part, as there is no universal solution in manufacturing.



OD Rougher

An OD rougher is a tool used for roughing operations on the outer diameter (OD) of a workpiece.

OD Finisher

An OD finisher refers to a tool used for finishing operations on the outer diameter (OD) of a workpiece.

OD Groover

An OD groover is a specialized tool used for creating grooves on the outer diameter (OD) of a cylindrical workpiece.

Face Groover

A face groover is a specialty tool that's used for creating grooves on the face of a workpiece.

Threader

A threader is a specialty tool used for cutting threads on the outer or inner surfaces of a cylindrical workpiece.

Drills and Taps

Drills are used to create holes in workpieces, while taps are used to cut internal threads in a pre-drilled hole.

ID Groover

ID groovers are used to create grooves on the inner diameter (ID) of a cylindrical workpiece.

ID Boring Bar

ID boring bars are specialty tools that enlarge and finish the inner diameter of a cylindrical part.

INFORMATION@HPPI.COM | 303-530-3131

CNC TURNING

Turning Operations

The turning process uses a lathe machine to move the cutting tool in a straight line along the rotating workpiece. This removes material around the edge until the desired diameter is reached, allowing for the creation of cylindrical parts with features like slots, tapers, and threads.

Straight Turning

Straight turning reduces the workpiece diameter evenly to ensure consistency. It is typically a "roughing" process that removes large amounts of material before precision cutting.

Facing

Facing, or face turning, is a method to create a flat surface at a right angle to the workpiece's rotation. The tool is secured in a holder on the lathe's carriage. It moves across the part's rotational axis. Face turning can be done as an initial rough cut or a final finishing cut.

Drilling

Drilling can be performed using a lathe or a turning center. Advanced turning centers allow drilling in various orientations, not just along the central axis.

Boring

Boring is used after drilling to make a hole larger. The tool enters the existing hole and removes material from its inner wall.

Turning Ops

INFORMATION@HPPI.COM | 303-530-3131

CNC TURNING

Turning Operations

Taper Turning

Taper turning creates a cylinder with a diameter that gradually decreases.

Grooving

Grooving is done when a shaped tool is pressed into the component gouge to form a narrow cavity. Grooving is typically used to create features like O-ring or circlip grooves and contouring operations. A grooving tool is sometimes also called a parting-off tool and is used to remove the completed part from the workpiece stock.

Parting

Parting involves cutting deeply into the material to separate the finished piece from the original stock.

Knurling

Knurling creates a diamond pattern on the outer surface of a part by compressing the material, not cutting it. This technique is primarily used to enhance grip on parts, especially those requiring extra traction.

Threading

Threading involves creating grooves in a hole or on the outer surface of a workpiece, allowing it to be screwed into other objects.

Turning Ops

Geometric Dimension and Tolerancing

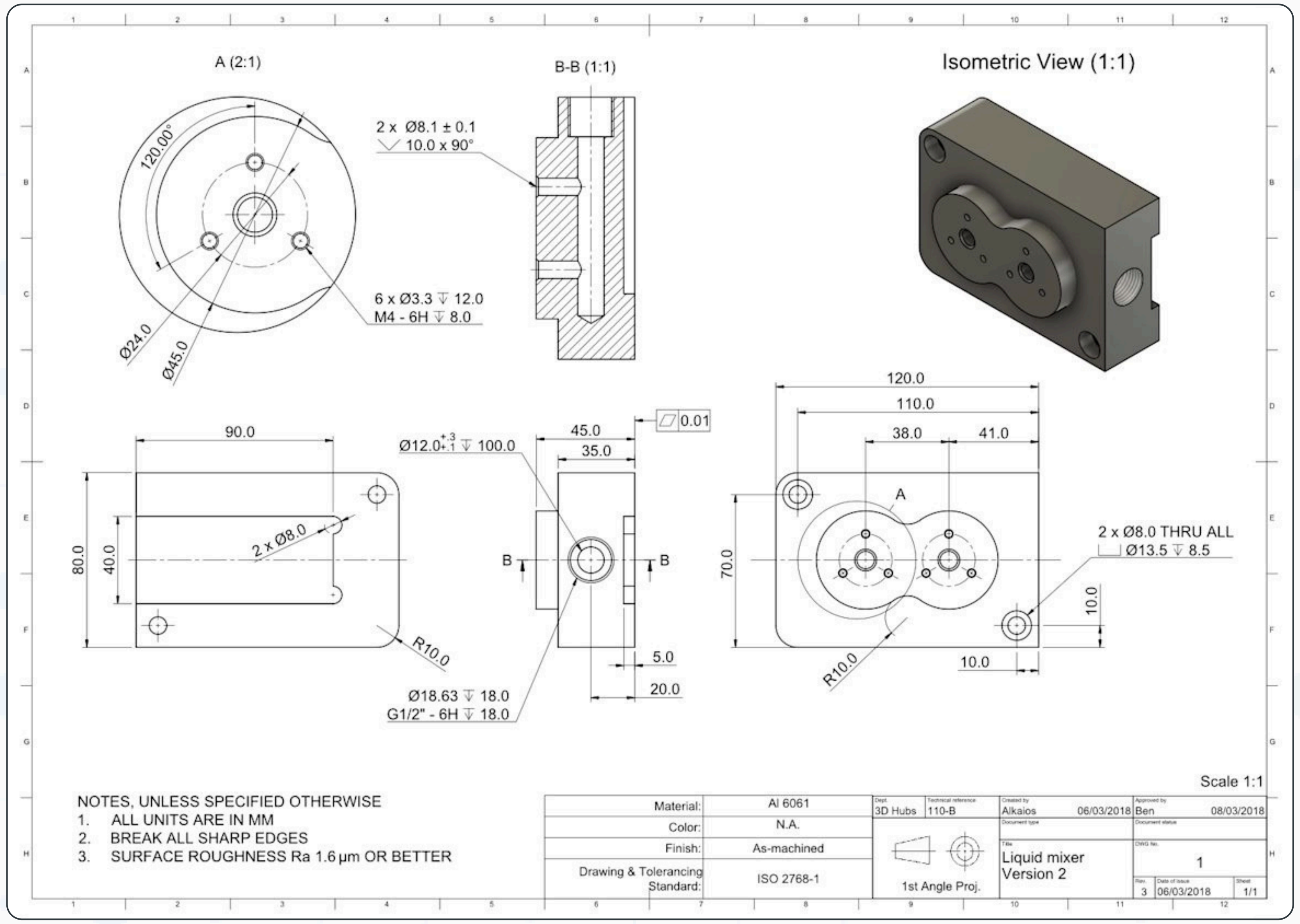
Geometric Dimensioning and Tolerancing (GD&T) provides a different method for defining dimensions and tolerances compared to traditional plus/minus tolerancing. While engineers create parts with ideal geometry in CAD, the reality is that manufactured parts are never flawless.

Using GD&T correctly enhances quality and cuts down on delivery time and costs. It does this by offering a universal language to clearly convey design intentions and by concentrating on functional interfaces for part tolerancing.

These are the main benefits of using GD&T:

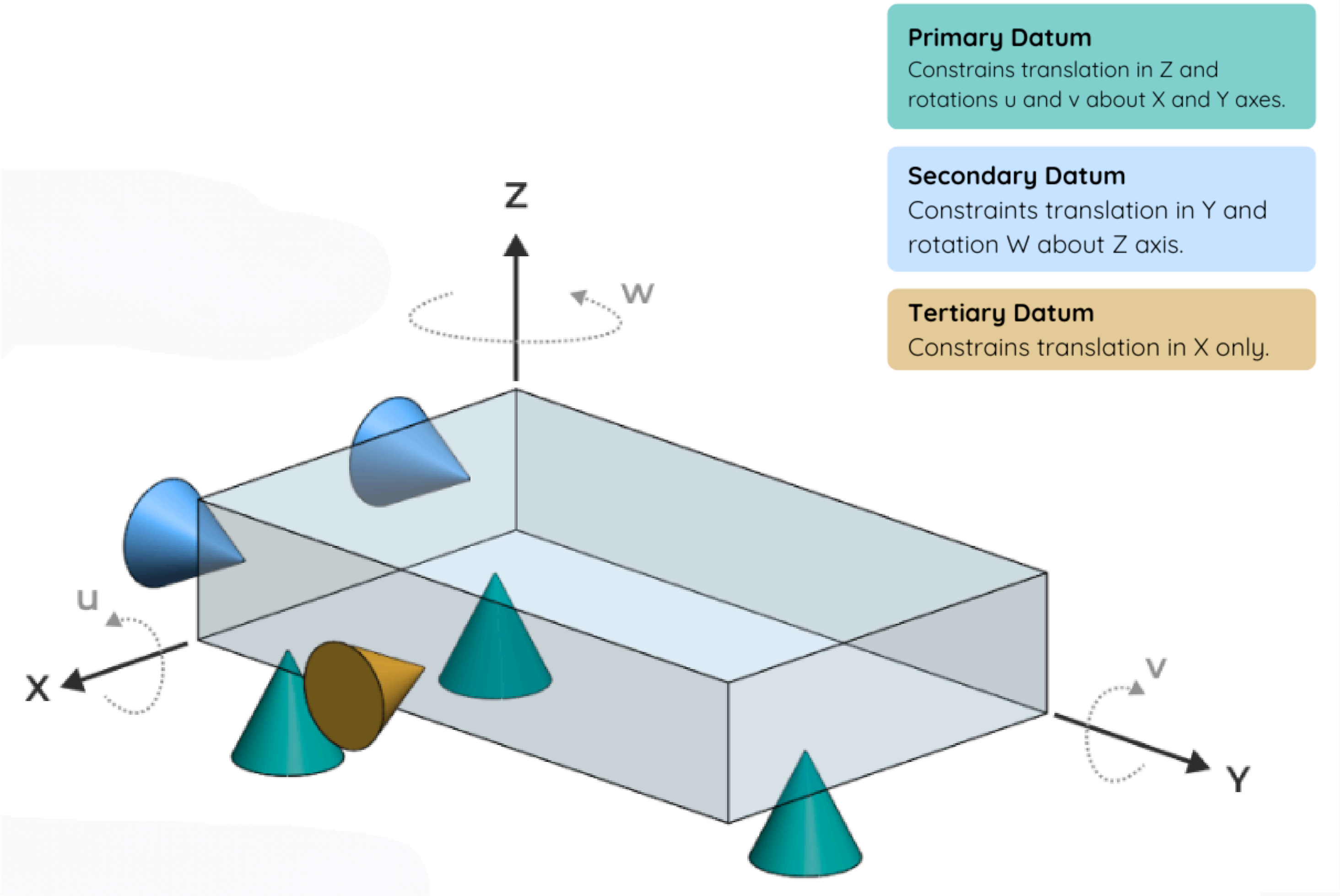
- Standardized design language
- Clear, precise, and consistent communication between customers, suppliers, and production teams
- Method for calculating the worst-case mating limits
- Repeatable production and inspection processes
- Assembly is assured from qualified production parts

A thorough understanding of GD&T will significantly improve communication with your manufacturing and quality control teams or supplier, increase the quality of your parts, and reduce costs.



GD&T

Datums and Features



A datum is a theoretical exact plane, axis, or point location that serves as a reference point for measurements on machined components. Datums ensure that parts are manufactured consistently and accurately.

Primary datum: This is the first datum feature used to establish the datum reference frame. It is the most critical datum and is usually chosen based on the function of the part in the assembly. The primary datum provides the main reference for all other measurements

Secondary datum: This datum feature is used in conjunction with the primary datum to further define the datum reference frame. It provides additional orientation and location references for the part

Tertiary datum: This is the third datum feature used to complete the datum reference frame. It provides the final orientation and location references, ensuring that the part is fully constrained in all degrees of freedom

Features, also known as datum features, are the actual physical features on the part that are used to establish the datums. They can be points, lines, planes, or a combination.

These features can include surfaces, edges, holes, slots, and other geometric elements that define the part’s shape and functionality.

GD&T

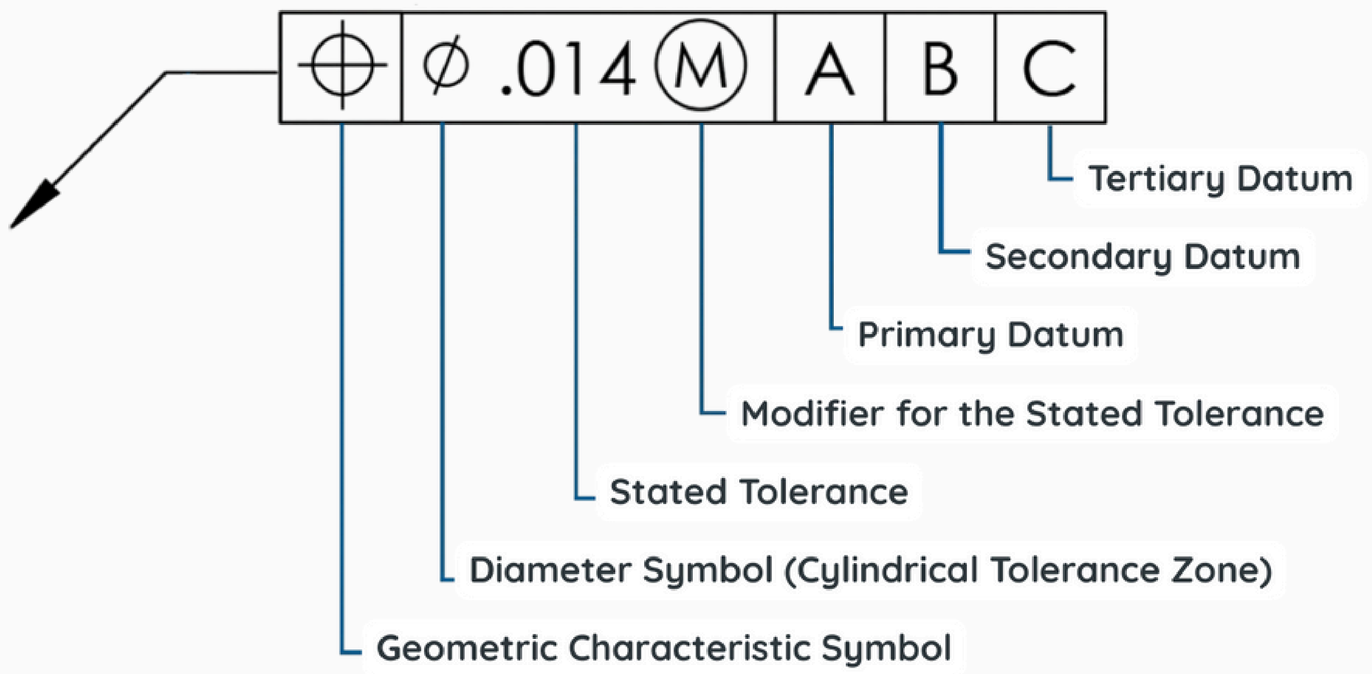
Feature Control Frame

The Feature Control Frame (FCF) is a rectangular box that contains the geometric characteristics and tolerance information for a feature on a part. It is a critical element in GD&T because it communicates how much variation is allowed in the geometry of a part feature and how that feature relates to other features.

Each frame can hold only one symbol. If a feature has two requirements, use either two separate frames or a composite tolerance. The symbol indicates the type of control applied to the feature.

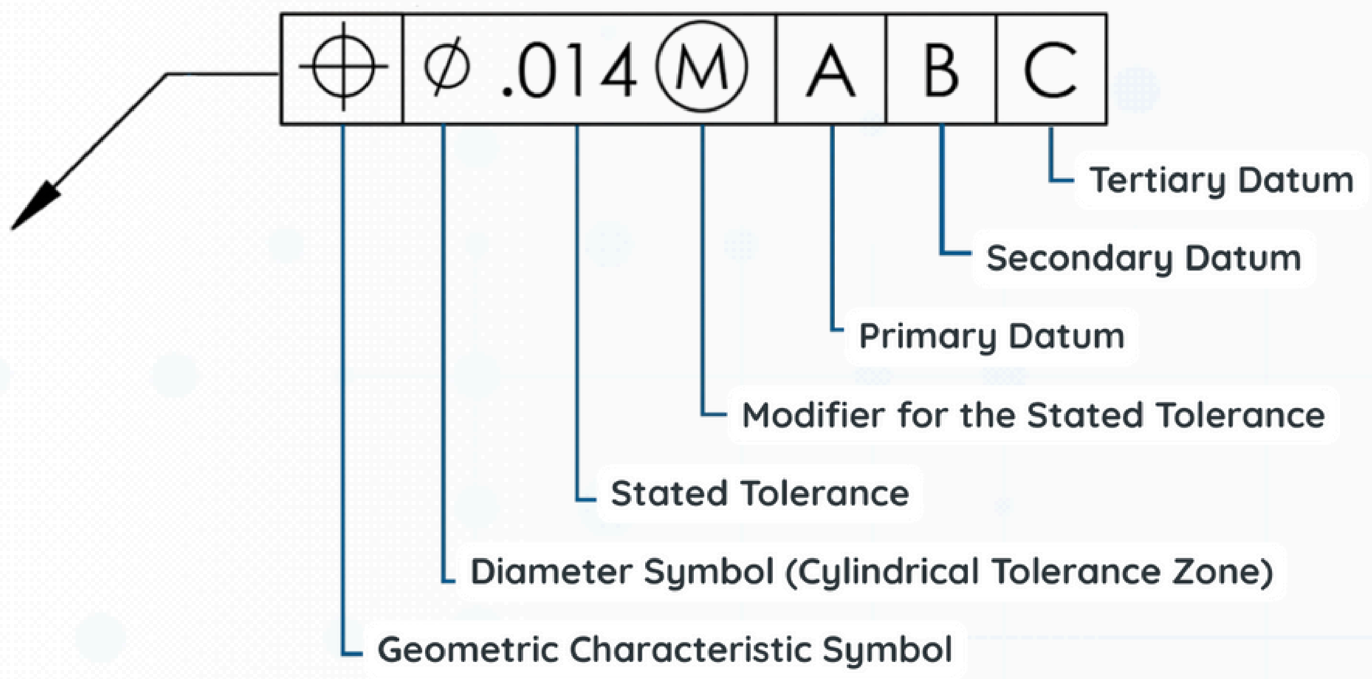
The second part of a feature control frame shows the complete tolerance for the feature. This tolerance is always a total value, not a plus/minus range.

If the tolerance has a diameter symbol (∅) before it, imagine the tolerance zone as a circular or cylindrical area—commonly used for hole positioning. Without this symbol, the tolerance zone defaults to parallel planes, typically used for positioning slots or surface profiles.



A typical Feature Control Frame is divided into compartments, each containing specific information:

- **Geometric Characteristic Symbol**: This is the first compartment and indicates the type of control being applied (e.g., flatness, position, perpendicularity).
- **Tolerance Value**: The second compartment shows the tolerance amount and any modifiers (like MMC – Maximum Material Condition).
- **Datum References** (if applicable): The following compartments list the datum features (A, B, C, etc.) that the controlled feature is referenced to, in order of precedence.



GD&T

Feature Control Frame

After the feature tolerance in the feature control frame, you might see a material condition modifier like Max Material Condition (MMC) or Least Material Condition (LMC) for features of size, such as holes. If no modifier is specified, the default is RFS (Regardless of Feature Size), though it's not shown in the frame. For non-size features, like plane surfaces, these modifiers aren't used.

The remaining sections of the feature control frame will include datum feature references if needed. For instance, if a form tolerance like flatness or straightness is specified, no datum reference is used. Conversely, if a location tolerance, such as position, is specified, datum references are typically included.

The order of datum references is based on their importance, not the alphabet. They are read from left to right as primary, secondary, and tertiary. Typically, Datum A is the primary, followed by B and C.

The primary feature is the first to make contact, requiring at least three points of contact. The secondary feature is the next, needing at least two points, and the tertiary feature is the last, with at least one point of contact.

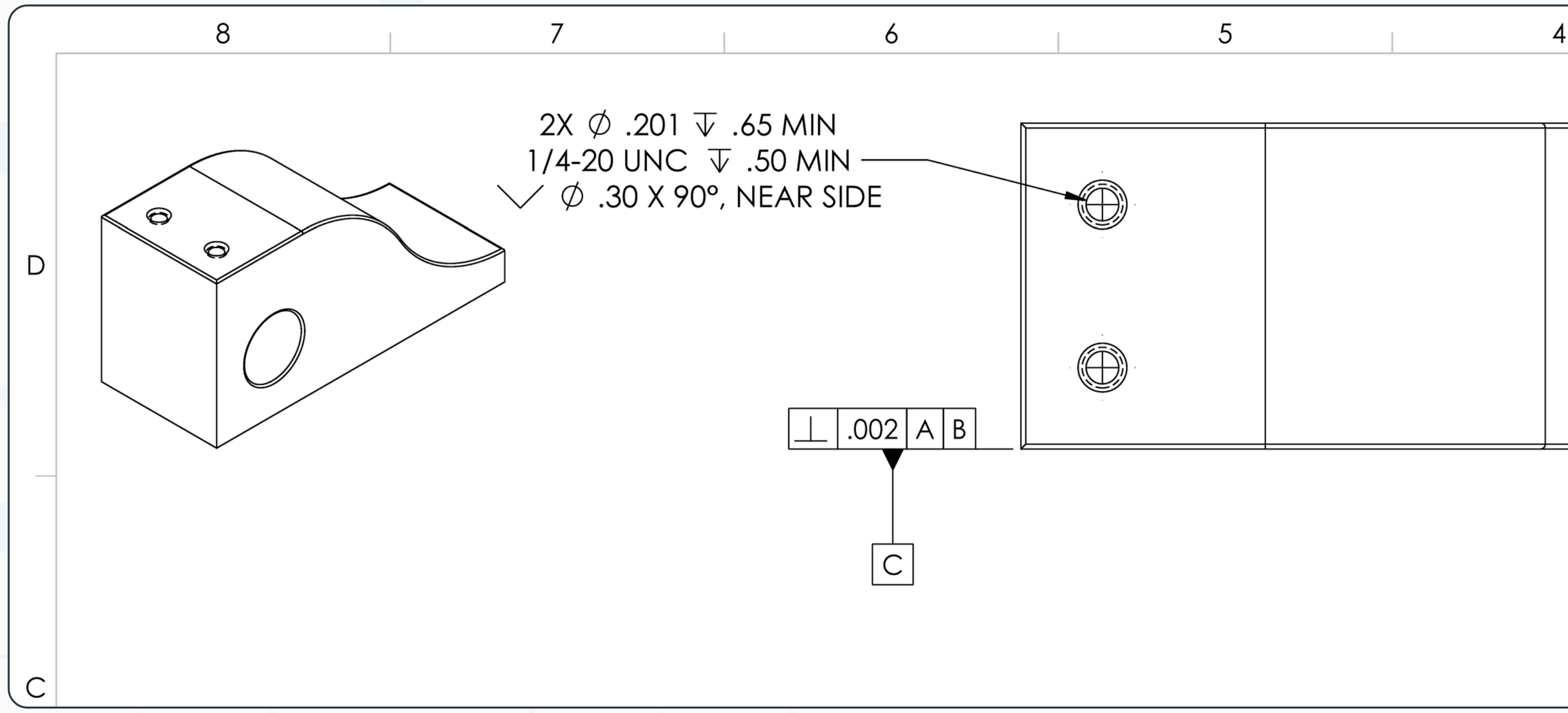
When all three datum features are contacted simultaneously, they form three perpendicular datum planes, known as the Datum Reference Frame (DRF). The DRF is established using Datum Simulators, which include manufacturing, processing, and inspection tools like a surface plate, collet, three-jaw chuck, or gage pin.

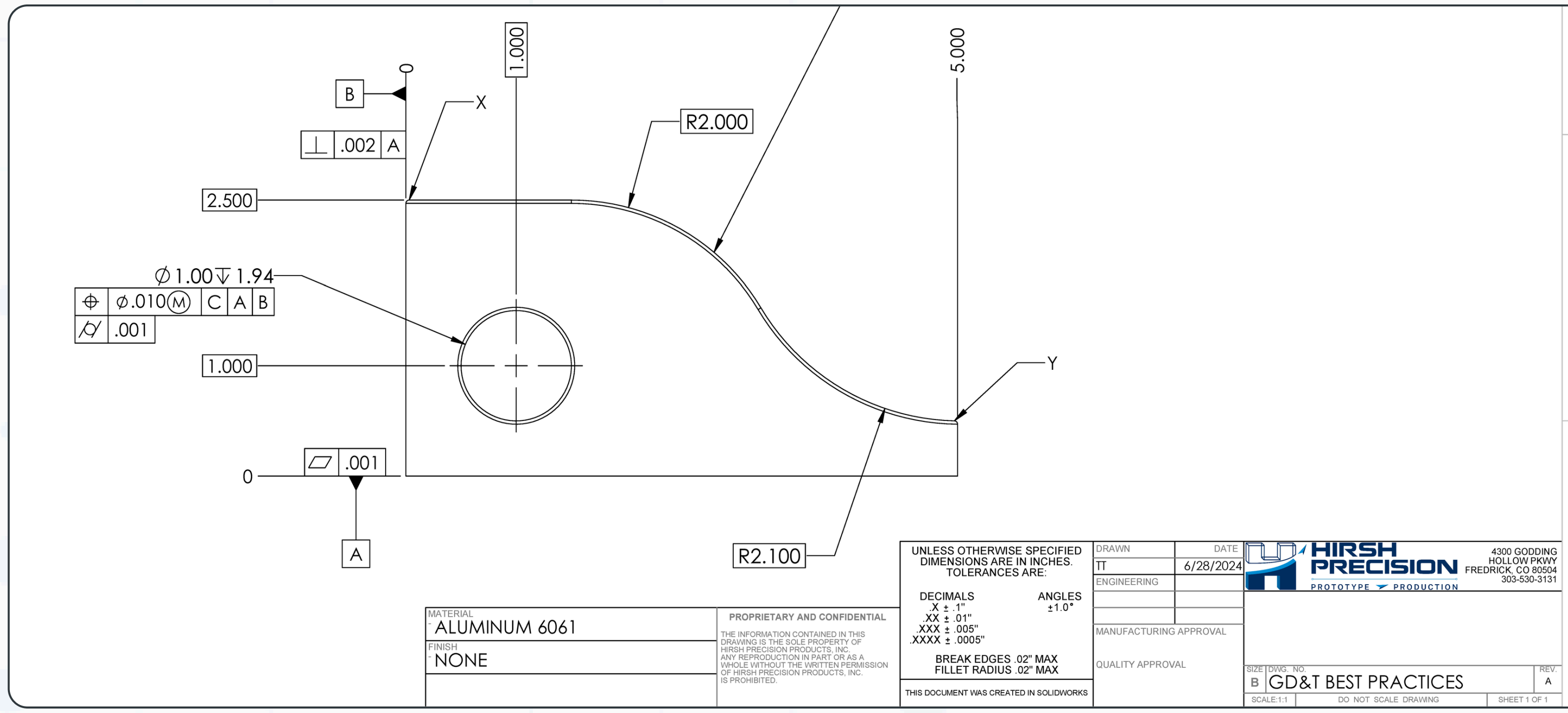
GD&T

Material Condition Modifiers

When defining geometric controls, it's important to specify that a tolerance is linked to a feature's size. Engineers use Maximum Material Condition (MMC) and Least Material Condition (LMC) to express this requirement.

These material condition modifiers are placed in a feature control frame after the feature tolerance. Using MMC and LMC modifiers allows for extra geometric tolerance, known as "bonus" tolerance, when features move away from the specified condition.





Maximum Material Condition (MMC)

The condition where a feature holds the most material possible within specified size limits, such as the largest pin or the smallest hole.

Least Material Condition (LMC)

The condition where the feature contains the least material within the stated limits of size. (ex: smallest pin and/or largest hole).

GD&T

Symbols

GD&T is a system where all engineered parts are made up of features. Geometric tolerances are set on these features using feature control frames, which use symbols to show the allowed tolerance.

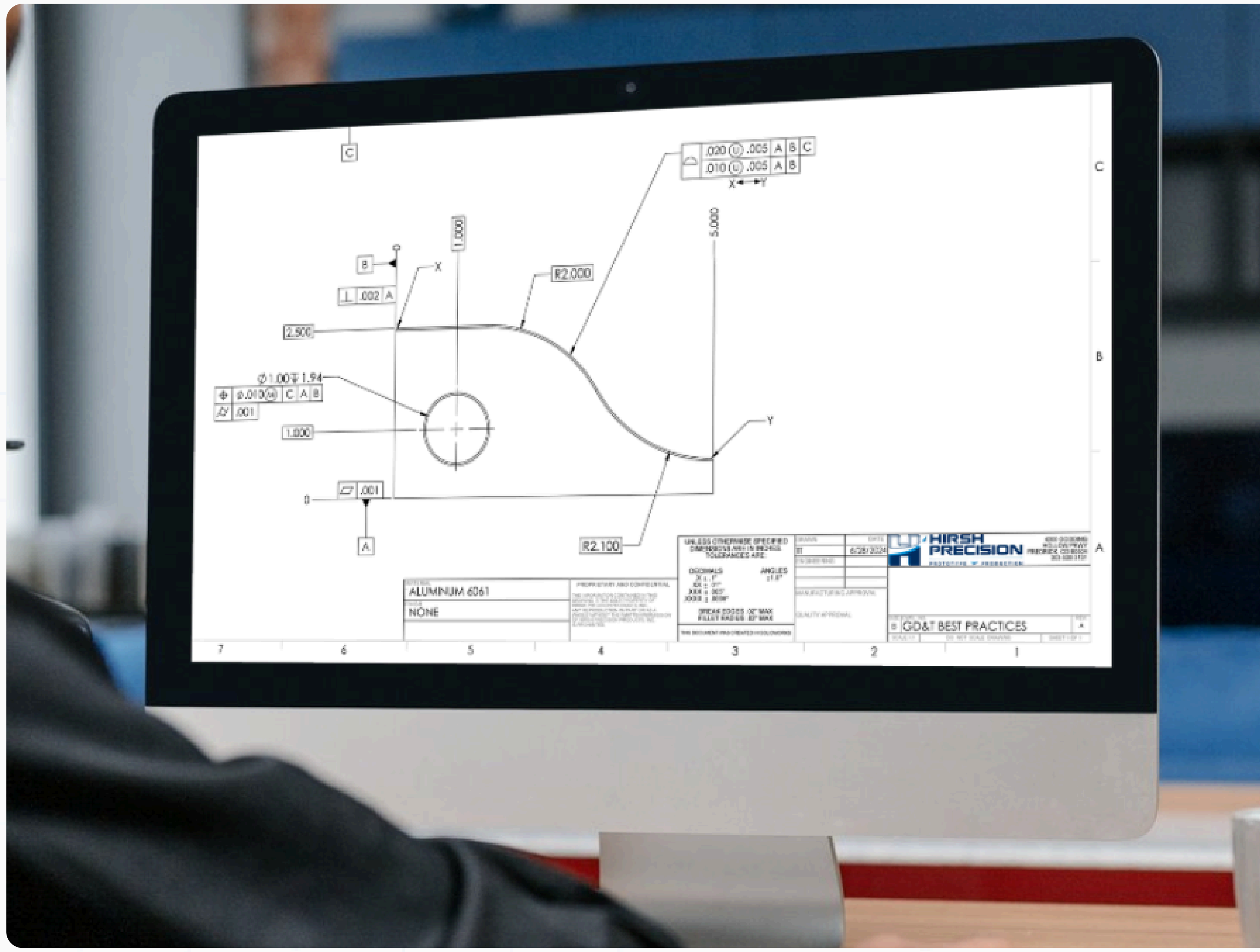
These characteristics and their symbols fall into four main categories (or characteristics of features): form, orientation, location, and runout.

Form tolerances control the “shape” of features and are often used as a refinement of size, which means they do not require a datum reference.

Orientation tolerances control the "tilt" of features, link to basic angle dimensions, and refine location. Because orientation GD&T is relative, these feature control frames always reference a datum. When applied to surfaces, orientation tolerances manage form.

Location tolerances control the location and are linked to basic linear dimensions. Location GD&T can position a feature or its size based on the feature itself or a set of derived median points. These characteristics are highly versatile and powerful, allowing control over size, form, and orientation within a single feature control frame.

Runout tolerances control the functional and rotational accuracy of a part feature, usually cylindrical or rotational parts, by limiting how much a surface or feature can deviate as it spins around a datum axis.



INFORMATION@HPPI.COM | 303-530-3131

GD&T

Symbols

Orientation tolerances control the "tilt" of features, link to basic angle dimensions, and refine location. Because orientation GD&T is relative, these feature control frames always reference a datum. When applied to surfaces, orientation tolerances manage form.

∠ Angularity

Controls the deviation of a line on a surface or an axis within a tolerance zone that is defined by two parallel lines a distance apart; no datum required.

// Parallelism

Controls parallelism between two parallel surfaces, with two parallel planes (both parallel to the datum feature) acting as the tolerance zone.

⊥ Perpendicularity

Controls perpendicularity between two 90-degree surfaces, with two parallel planes (both perpendicular to the datum feature) serving as the tolerance zone.

INFORMATION@HPPI.COM | 303-530-3131

GD&T

Symbols

Location tolerances control the location and are linked to basic linear dimensions. Location GD&T can position a feature or its size based on the feature itself or a set of derived median points. These characteristics are highly versatile and powerful, allowing control over size, form, and orientation within a single feature control frame.

Position

Also known as True Position, it controls how far a feature of size can deviate from where it should be; the acceptable area is generally defined by a circular or cylindrical tolerance zone.

Surface Profile

A 3D tolerance zone that defines where the surface needs to be located; it can also be applied to curved surfaces.

Line Profile

A 2D tolerance zone that defines the profile along a 2D cross section of a surface.

INFORMATION@HPPI.COM | 303-530-3131

GD&T

Symbols

Runout tolerances control the functional and rotational accuracy of a part feature, usually cylindrical or rotational parts, by limiting how much a surface or feature can deviate as it spins around a datum axis.

Total Runout

Controls the acceptable variation in a surface when it's rotated 360 degrees around a central axis (the datum feature); the tolerance zone is defined by two concentric cylinders.

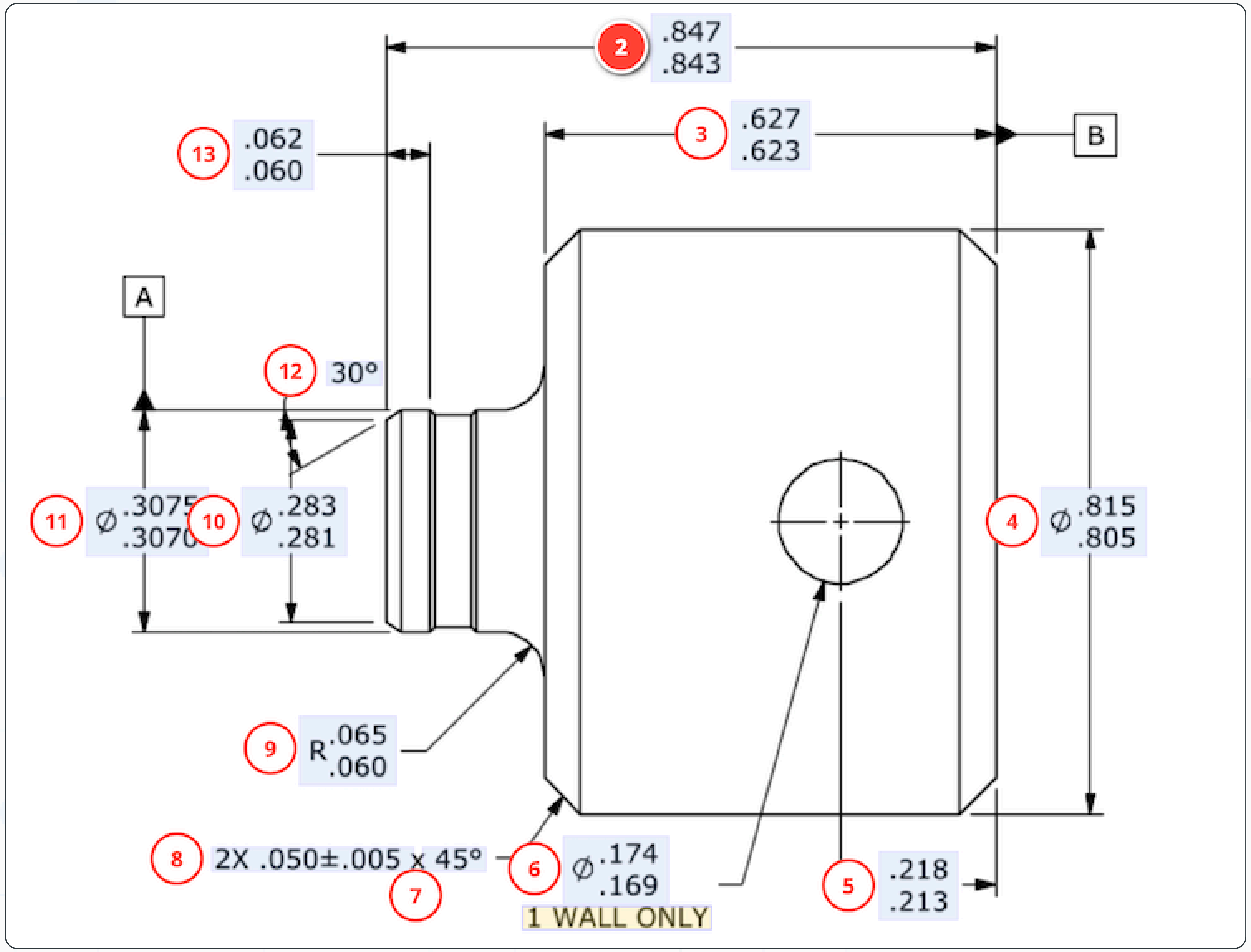
Circular Runout

The 2D version of Total Runout.

Tolerances

Tolerances refer to the permissible range of variation in a dimension that ensures a part fits and functions correctly. According to ASME, engineering tolerance is the total allowable deviation from a dimension's basic value.

Tolerances are checked during the quality inspection after CNC machining. They guide the choice of manufacturing methods needed for a part. Tighter tolerances might need advanced equipment or extra processing to meet the required dimensions.
For CNC machined parts, various types of tolerances can be defined and used.



Related Read: 10 GD&T Best Practices

INFORMATION@HPPI.COM | 303-530-3131

TOLERANCES

Types of CNC Machining Tolerances

In CNC machining, "tolerance" refers to how precisely a machine can shape a part. CNC machines are incredibly precise, with some achieving an accuracy of ± 0.0025mm, about a quarter of a human hair's width. However, tolerances differ among machines and are typically set by the manufacturer, with 0.02mm being a common standard. CNC service providers also inform customers about their machines' tolerances.

In design and manufacturing, tolerance refers to the allowable range of size variations for a part that still ensures it works properly. Designers set these tolerances based on the part's function, fit, and form. Tolerances are vital for parts that connect or interact with others. For instance, parts of an electric engine require tighter tolerances than a door handle because they have more features that need to fit together precisely. Tolerances are indicated by a number next to the relevant dimension.

Standard Tolerances

Standard tolerances are widely used to define how much a CNC machined part's dimensions can vary. These tolerances might represent what a machine shop can produce without extra cost or serve as a general variation applied to an entire drawing, as noted in a title block or note, unless specified otherwise in the drawing or on certain features.

Limit Tolerances

A limit tolerance is the specified minimum and maximum dimension allowed on a drawing. Also known as limits of size, these provide the upper and lower size limits for a feature. This makes it easy to quickly check if a measured dimension falls within the tolerance zone, without needing to calculate boundary dimensions from tolerance values.

Unilateral Tolerances

Unilateral refers to something related to one side. In terms of tolerances, it means the tolerance is applied in only one direction, either plus or minus. In GD&T, this is called an unequally disposed tolerance and is marked with a U symbol. Typically, unilateral tolerances are used at maximum material condition (MMC).

Bilateral Tolerances

Bilateral refers to something involving two sides. In terms of tolerances, it means that both the positive and negative limits are considered for a specific measurement. This provides the smallest and largest acceptable sizes for that measurement. The tolerance can be equal on both sides or vary between them.

INFORMATION@HPPI.COM | 303-530-3131

TOLERANCES

Sources of Variation

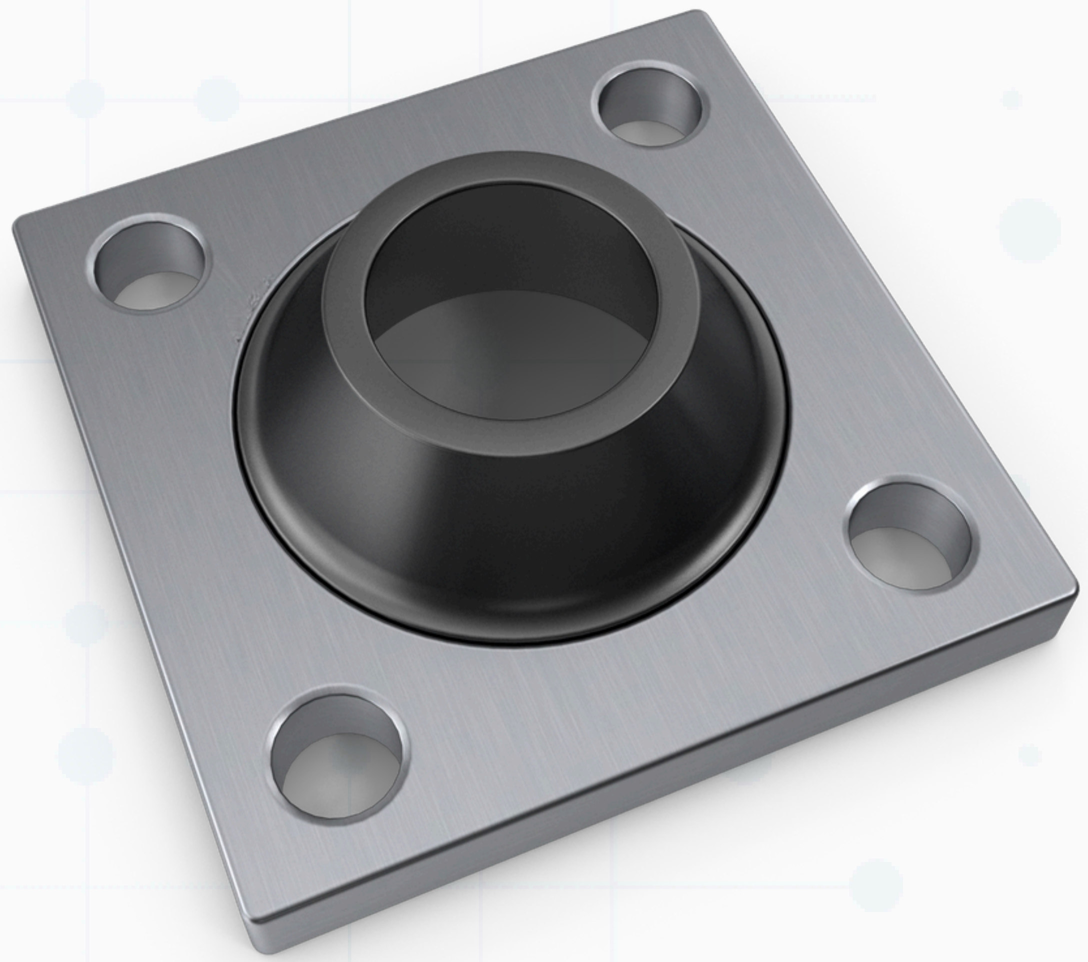
CNC-machined parts are not flawless. Variations among these parts are expected and result from several factors.

Raw Material
Harder materials allow for tighter tolerances because they maintain their shape and resist deflection during machining. They offer greater dimensional stability compared to softer materials. Additionally, materials with a lower coefficient of thermal expansion (CTE) are less affected by temperature changes, enabling even tighter tolerances.

Workpiece Condition
Using uniform and precise stock materials significantly minimizes variations. The quality of your raw materials directly affects the quality of your final product.

Machine Used
While more expensive machinery doesn't always guarantee higher quality parts, it often does. Machines that maintain tighter tolerances are typically built with superior raw materials and advanced programming, ensuring precise tooling alignment during milling.

Tooling
Without precise and accurate tooling, achieving tight tolerances is impossible, regardless of how well you prepare your material.



Material Selection

Broadly speaking, the material selection process for your CNC machining project should follow these steps:

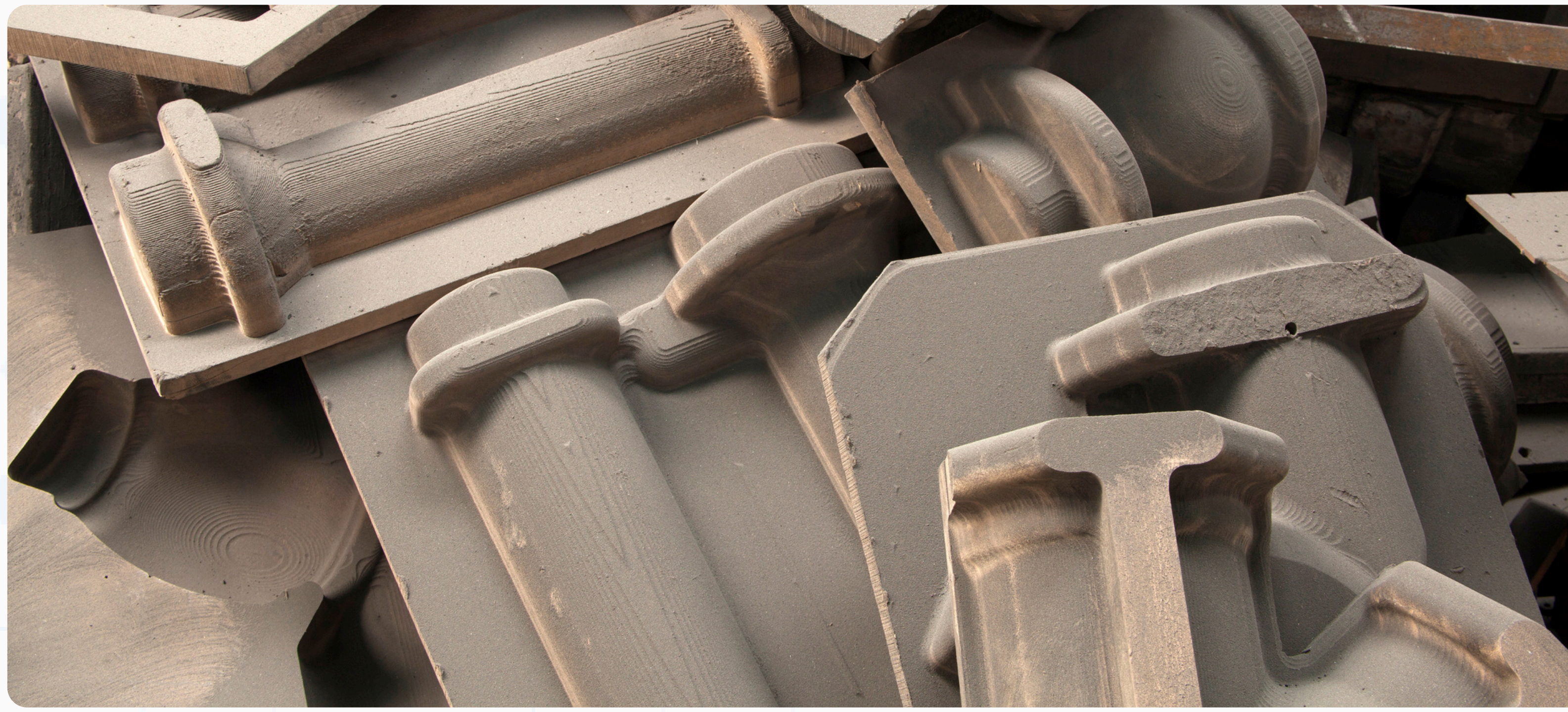
Step 1: Determine material requirements
Evaluate the material's functionality, electrical properties, strength, and hardness to ensure it fits your project needs. Additionally, consider the environment where the part will be used and the conditions it will face.

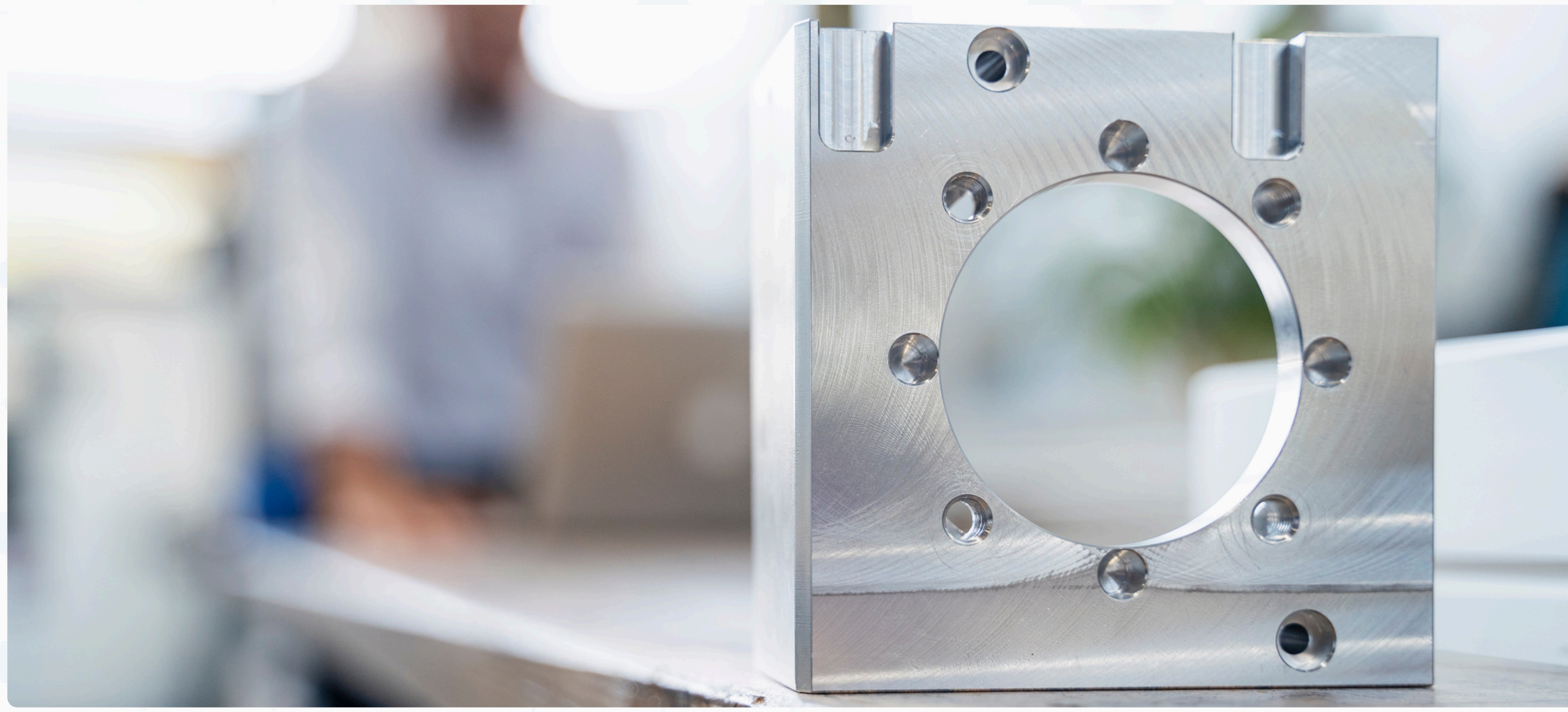
Step 2: Create a list of material options
Organize all the materials that meet your requirements, including your design specifications.

Step 3: Choose the most appropriate material
Select the material that meets most of your needs. Sometimes, you may need to prioritize a material with better machinability over a cheaper option to maintain part quality.

Design and regulatory requirements are the basis of your material selection process, but they can leave you with several material options to choose from. To make the best choice, consider the material's properties, machining needs, finishing requirements, and procurement factors.

By thoroughly evaluating these factors, you can identify the most cost-effective material that guarantees high-quality machining and reduces unnecessary delays in your operations.



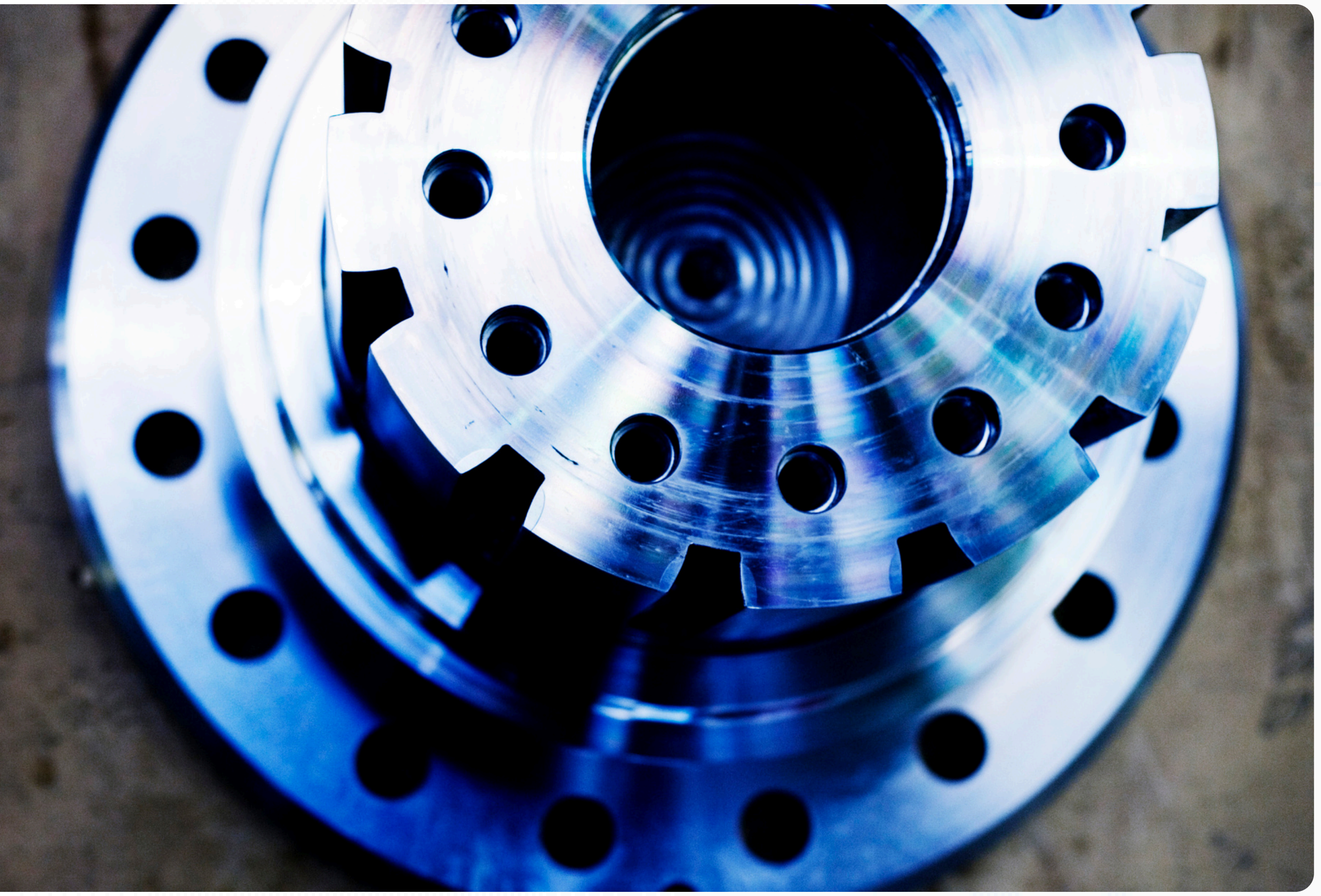


INFORMATION@HPPI.COM | 303-530-3131

MATERIAL SELECTION

Project Requirements

As it relates to your project, here are the key factors you need to consider when selecting the appropriate material.



Part Application

Choosing the right material for CNC manufacturing is crucial, and different applications require different materials.

The material's physical properties, like tensile strength, strength-to-weight ratio, crack resistance, rigidity, or flexibility, depend on its application. Heavier materials can handle more stress, but for weight-sensitive uses, choose lighter materials with a strong strength-to-weight ratio.

For example, aerospace parts must be lightweight, unlike parts for structural support. Aluminum 3.3211 is often used in aerospace because it has a strong strength-to-weight ratio.

Machining Conditions

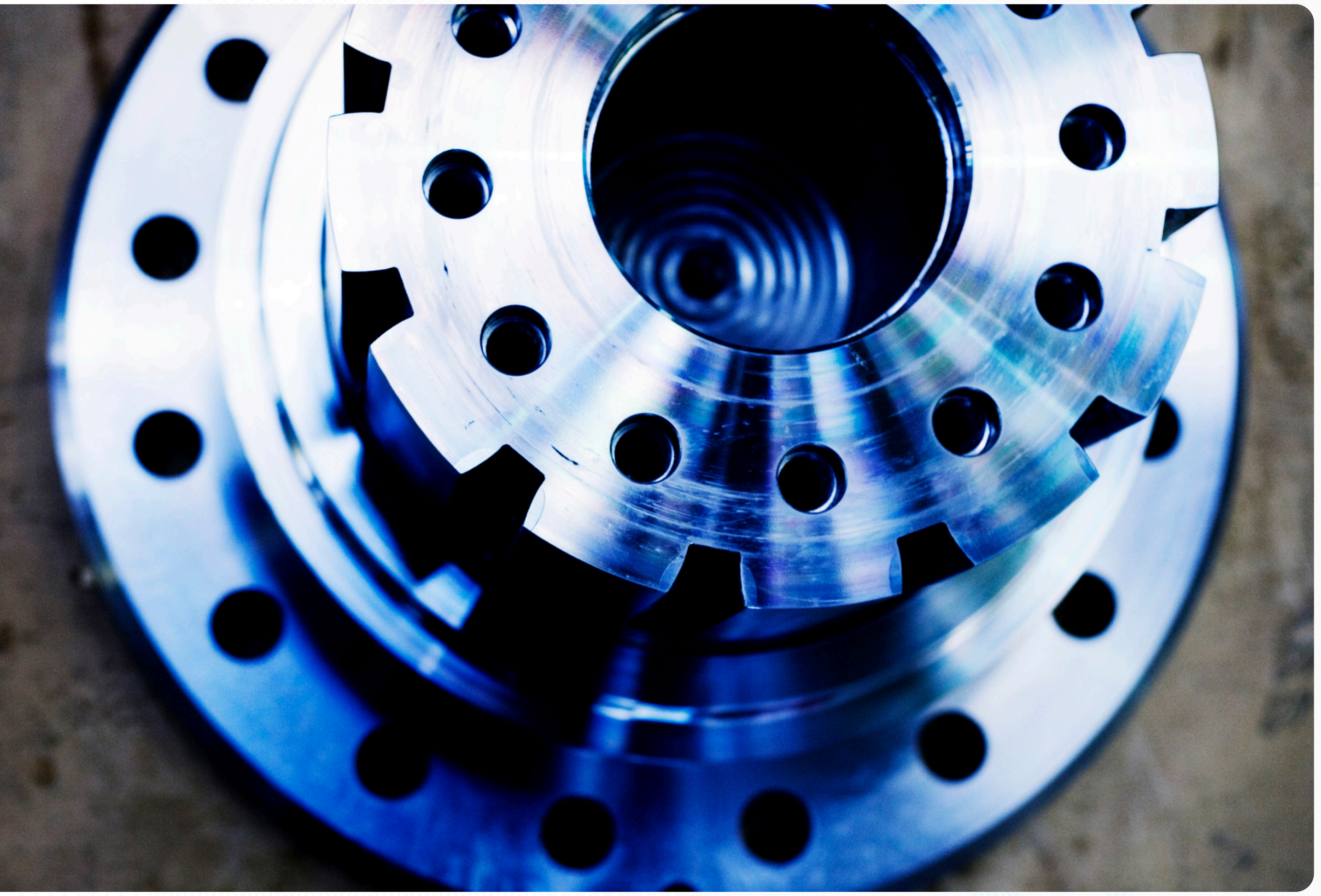
Finishing Requirements

Finishing Requirements

MATERIAL SELECTION

Project Requirements

As it relates to your project, here are the key factors you need to consider when selecting the appropriate material.



Part Application

Machining Conditions

The final application of your part will guide many of your material choices, but first, the raw material must undergo the machining process.

Machining rates, also known as speeds and feeds, determine how fast material is removed. Each material has specific machinability traits, like hardness and melting point, that affect how quickly it can be processed, how heat is handled, and how chips are cleared.

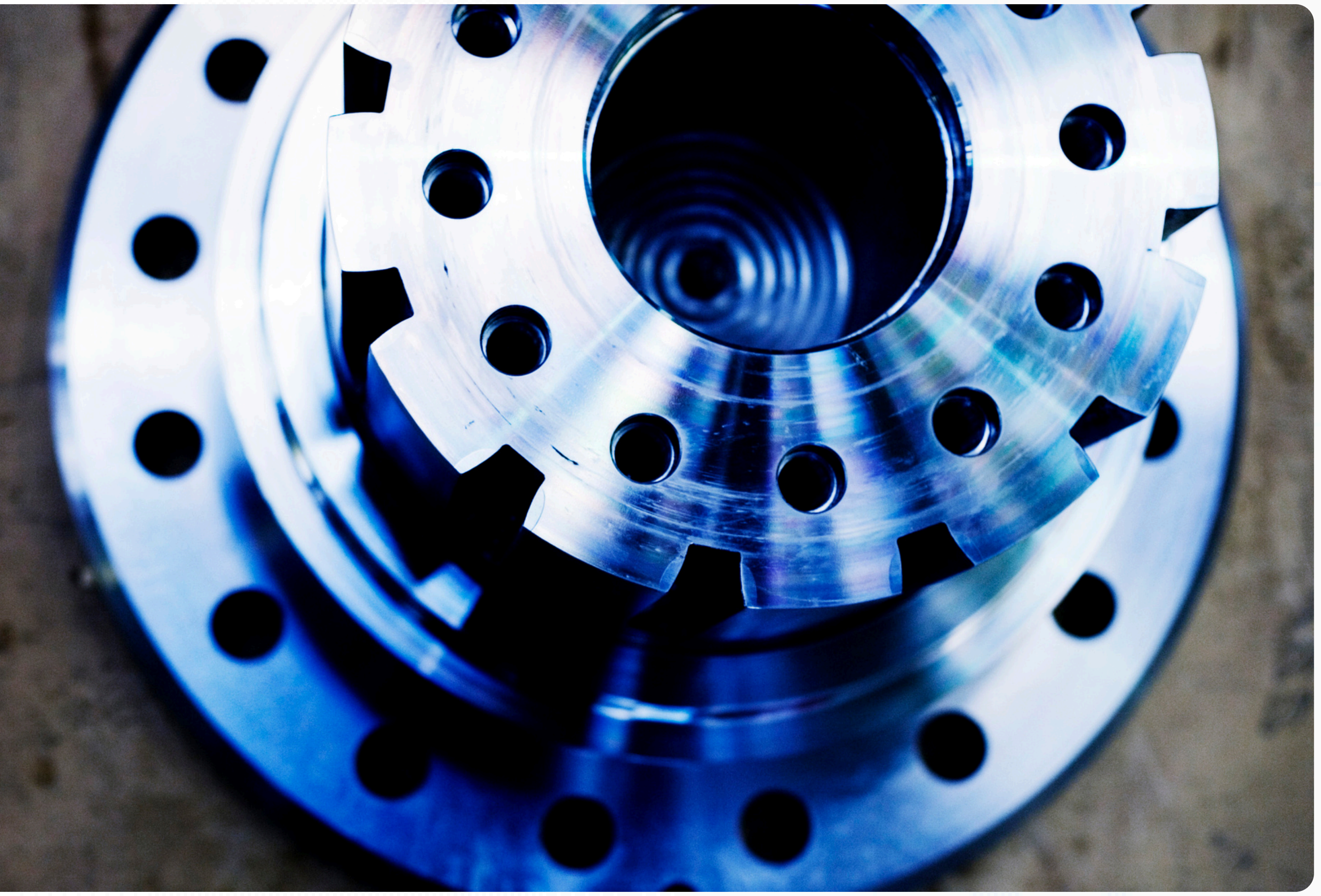
The material you choose impacts how quickly your cutting tools wear out. Some materials, like high-temperature alloys, can cause more wear, requiring special tools and increasing costs.

Finishing Requirements

MATERIAL SELECTION

Project Requirements

As it relates to your project, here are the key factors you need to consider when selecting the appropriate material.



- Part Application
- Machining Conditions
- Finishing Requirements

Don't be fooled by the word "finishing;" you need to carefully consider your parts' finish early in the design process, not as an afterthought. This helps avoid compatibility issues between your chosen material and the necessary surface process.

If your project requires a specific surface finish, choose a material that can be easily machined to meet those needs. Materials like aluminum and plastics can achieve a smooth finish effortlessly, while others, like cast iron, may require additional steps to reach the desired quality.

Check if the plating type matches the base material. For example, if nickel plating is needed, aluminum might not be the best choice. Instead, use stainless steel, which works well with nickel plating, ensuring you select the right material.

INFORMATION@HPPI.COM | 303-530-3131

MATERIAL SELECTION

Material Attributes

Your part will have specific applications and environmental conditions that need to be taken into account when selecting a material.

For instance, parts for electrical use should conduct electricity well, like copper or silver. If used in aeronautics or aerospace, the material should maintain its shape under stress.

Tensile Strength
The maximum stress a material can withstand while being stretched or pulled before breaking. Materials with high tensile strength are essential for parts that will experience significant loads.

Hardness
A material's resistance to deformation, scratching, or indentation. Harder materials are more wear-resistant but may be more challenging to machine.

Ductility
A material's ability to deform under tensile stress, often characterized by its ability to be stretched into a wire. Ductile materials are less likely to fracture under stress.

Impact Resistance
A material's ability to withstand sudden and forceful impacts without breaking or shattering.

Attribute Types

Mechanical Attributes

Thermal Attribute

Electrical Attribute

INFORMATION@HPPI.COM | 303-530-3131

MATERIAL SELECTION

Material Attributes

Your part will have specific applications and environmental conditions that need to be taken into account when selecting a material.

For instance, parts for electrical use should conduct electricity well, like copper or silver. If used in aeronautics or aerospace, the material should maintain its shape under stress.

Thermal Conductivity
A material's ability to conduct heat. Materials with high thermal conductivity are essential for applications where heat dissipation is crucial.

Thermal Expansion
The degree to which a material expands when heated. Materials with low thermal expansion are preferred for applications requiring high dimensional stability across temperature changes.

Heat Resistance
A material's ability to maintain its properties at elevated temperatures. Materials with high heat resistance are necessary for high-temperature applications.

Attribute Types

Mechanical Attributes

Thermal Attribute

Electrical Attribute

INFORMATION@HPPI.COM | 303-530-3131

MATERIAL SELECTION

Material Attributes

Your part will have specific applications and environmental conditions that need to be taken into account when selecting a material.

For instance, parts for electrical use should conduct electricity well, like copper or silver. If used in aeronautics or aerospace, the material should maintain its shape under stress.

Electrical Conductivity

A material's ability to conduct electricity. Materials with high electrical conductivity are used in applications requiring efficient electrical transmission.

Dielectric Strength

The maximum electric field a material can withstand without breaking down. Materials with high dielectric strength are essential for insulating applications.

Electrical Conductivity

The measure of how strongly a material opposes the flow of electric current. Materials with high resistivity are used in applications requiring electrical insulation.

Attribute Types

Mechanical Attributes

Thermal Attribute

Electrical Attribute

INFORMATION@HPPI.COM | 303-530-3131

MATERIAL SELECTION

Procurement Factors

While design and industry regulations are crucial, consider these procurement factors when selecting a material for your machined component, which can help reduce unnecessary costs and affect your lead times.

Raw Material Cost

The cost of materials is a key factor in procurement. High-end aerospace alloys can be expensive, but more affordable options like plastics are available.

Machining Cost

The machining expenses for each material should be taken into account as well. Certain materials may demand additional time and labor to machine effectively, thus affecting the overall cost of machining.

Availability

When selecting materials, consider their availability. Some materials might be hard to find or take longer to get, which could delay your project. Make sure the material you choose is available within your required timeframe.

Lead Time

Consider the project's timeline as well, as the lead time for certain materials can significantly affect your material selection process. Some materials may have longer lead times than others, potentially causing delays in your overall project schedule.

INFORMATION@HPPI.COM | 303-530-3131

Post-Processing

Post-processing operations enhance material strength, add anti-corrosive properties, and improve surface smoothness. The most common post-machining processes are heat treatments and surface treatments.

Using the right heat treatment methods and surface finishes enhances the smoothness, appearance, and functionality of CNC machined parts.

Heat Treatment Methods

Heat treatment is a crucial step in CNC machining metal parts. It involves carefully heating and cooling the material to achieve specific properties. This process changes four key properties of metal parts: hardness, strength, toughness, and ductility.

Not all heat treatments are the same. The right method depends on the material's composition, the part's size, and the desired final properties.

Heat treatment aims to create a specific microstructure that imparts desired material properties to your parts, and can be applied to metal alloys at different stages of the CNC machining process, either before or after machining your parts.

Surface Finishing Methods

Surface finishing methods protect and enhance the look of a machined component's surfaces. These methods may add or remove material, or use heat, electricity, or chemicals to alter a part's surface finish.

CNC surface finish and finishing methods are crucial when your part interacts with other components. For instance, ball bearings are designed to minimize rotational friction and support loads. As the races rotate, the balls also turn due to their contact. If the surfaces of the balls or races are not finished correctly, friction increases, leading to more wear and a shorter lifespan, even if the components meet geometric tolerances.

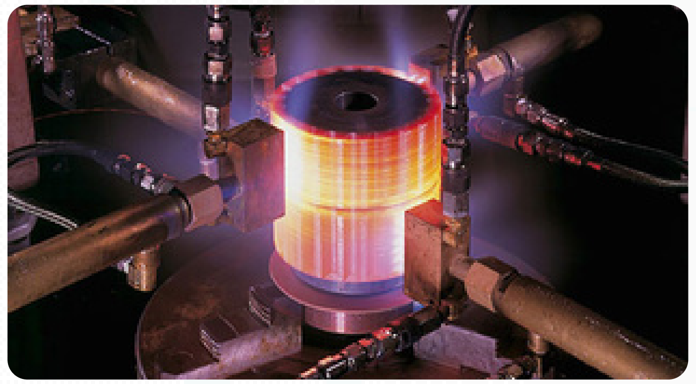
POST-PROCESSING

Heat Treatment Methods

Annealing

Annealing involves heating metal to a high temperature and cooling it slowly to achieve the desired structure. It's used on metal alloys after they are formed and before any further processing to make them softer and easier to machine. Unless specified otherwise, most CNC machined parts will have the properties of annealed material.

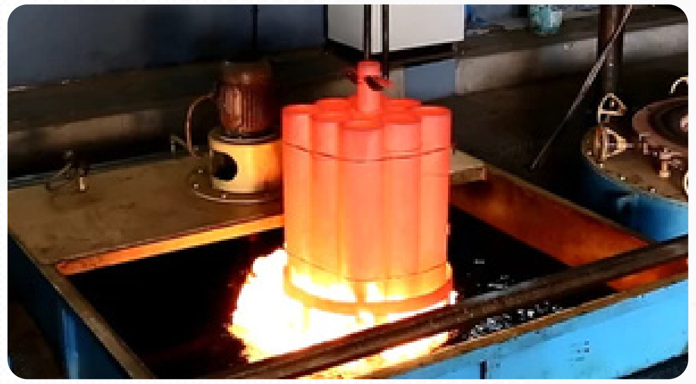
Purpose: improve the machinability of the metal alloy
Compatible Materials: steel, stainless steel, cast iron



Tempering

Tempering involves heating the part at a lower temperature than annealing, typically after quenching mild steels (1045 and A36) and alloy steels (4140 and 4240). This process reduces brittleness and enhances mechanical performance.

Purpose: release residual stress from mechanical deformation or temperature increase during processing
Compatible Materials: steel, stainless steel, cast iron



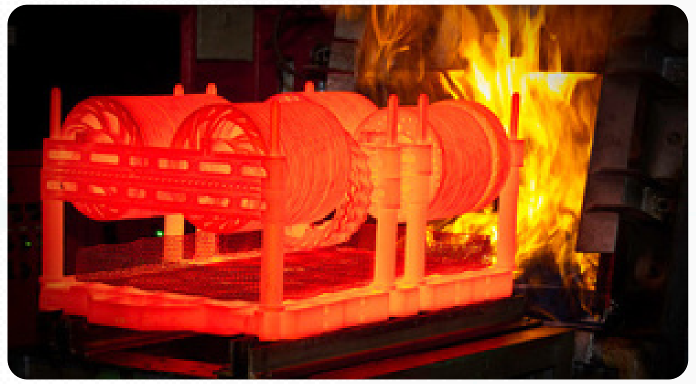
Quenching

Quenching involves heating metal to a high temperature and then quickly cooling it, usually by immersing it in oil or water, or by using cool air. This rapid cooling solidifies the changes in the metal's structure, making it extremely hard.

Purpose: increase the hardness of the metal alloy
Compatible Materials: mild steels, alloy steels, tool steels

POST-PROCESSING

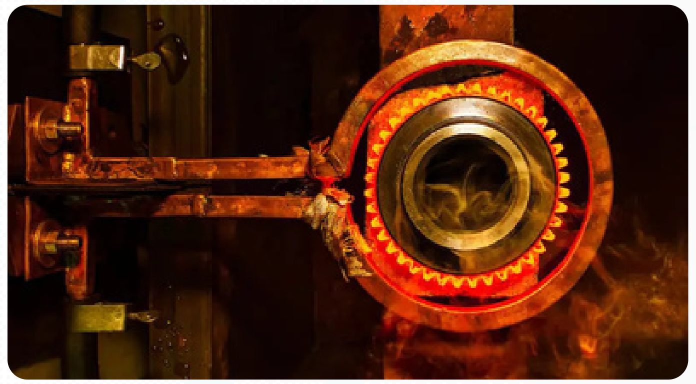
Heat Treatment Methods



Stress Relieving

Stress relieving heats the part to a high temperature, lower than annealing, and is typically done after CNC machining to remove residual stresses from manufacturing. This process ensures parts have more consistent mechanical properties.

Purpose: reduce brittleness after quenching
Compatible Materials: mild steels, alloy steels, tool steels



Case Hardening

Case-hardening, also known as carburizing, is a heat treatment that strengthens the metal's surface while keeping the inner metal soft. This process involves adding carbon or nitrogen to low-carbon alloys at high temperatures to enhance their hardenability. The surface finish can be applied either before or after the CNC machining process.

Purpose: improve surface toughness and increase corrosion resistance
Compatible Materials: steel alloys, copper, aluminum, brass

Through Hardening

Through hardening is different from case hardening because it strengthens the entire alloy, not just the surface. This is done by adding carbon to the alloy and cooling it quickly in brine, water, or oil.

Purpose: enhance the wear resistance and fatigue strength
Compatible Materials: alloy steels, tool steels, stainless steels

INFORMATION@HPPI.COM | 303-530-3131

POST-PROCESSING

Surface Finishing Methods

Media Blasting

Media blasting uses compressed air or water to shoot abrasive materials, known as "media," at high speeds onto a surface. This process cleans and prepares the surface for further treatment or finishing.

The type of abrasive material chosen for media blasting varies based on the task and the results you want to achieve.

Purpose: to achieve a smoother surface
Compatible Materials: most metals and plastics

Black Oxide

Black oxide is a coating applied by dipping a part into a sodium hydroxide and potassium nitrate solution. This process creates a black, smooth finish that enhances corrosion and wear resistance.

Purpose: to improve corrosion and wear resistance

Compatible Materials: mild steels, carbon steels, stainless steel, copper

Options

INFORMATION@HPPI.COM | 303-530-3131

POST-PROCESSING

Surface Finishing Methods

Powder Coating

Powder coating involves using powdered plastic mixed with chemical agents like pigments and additives to create a strong finish on metal parts.

The powder is sprayed onto a part and sticks due to an electrostatic charge. The coating tool charges the powder particles while the part is grounded, drawing the particles to it. The part is then baked, turning the powder into a solid, durable, and corrosion-resistant coating. This process allows for various pigments and finishes. Sometimes, parts are preheated before spraying.

Purpose: to increase strength and provide a good base for dyeing
Compatible Materials: aluminum, stainless steel, steels, copper, brass, zinc

Electroless Nickel Plating

Electroless nickel plating is a technique that adds a nickel layer to a part's surface. This enhances the part's corrosion and wear resistance, as well as increases its hardness.

Purpose: a cost-effective way to improve surface finish
Compatible Materials: aluminum, stainless steel, mild steels, copper, brass, titanium

Options

INFORMATION@HPPI.COM | 303-530-3131

POST-PROCESSING

Surface Finishing Methods

Electropolishing

Electropolishing is a process used to improve the surface finish of metal parts. It involves the removal of a thin layer of material from the surface of a metal object using an electrochemical process.

Purpose: to achieve an even, shiny surface that's corrosion resistance and highly weldable
Compatible Materials: stainless steel, aluminum, copper, brass, nickel alloys

Passivation

Passivation treats metal surfaces with an acid solution to remove free iron and contaminants, restoring a thin, protective oxide layer. This layer shields the metal from environmental reactions, reducing corrosion risk. It is often used on stainless steel and other metals to enhance resistance to rust, stains, and deterioration.

Purpose: to improve corrosion resistance
Compatible Materials: stainless steel, aluminum, titanium

Options

Media Blasting

Black Oxide

Powder Coating

Electroless Nickel Plating

Electropolishing

Passivation

Anodizing

Galvanizing

INFORMATION@HPPI.COM | 303-530-3131

POST-PROCESSING

Surface Finishing Methods

Anodizing

Anodizing creates a thin layer on metal surfaces, offering protection against corrosion and wear. It only works with aluminum and titanium.

In Type II and Type III anodizing, the part is placed in a diluted sulfuric acid solution, and an electric voltage is applied. This causes a reaction that turns the surface into hard aluminum or titanium oxide. Masks can be used on areas that need to stay conductive or have precise dimensions, like threaded holes, to prevent anodizing. The anodized parts can also be dyed in various colors before sealing.

Type II anodizing, also known as "standard" or "decorative" anodizing, creates coatings up to 25 μm thick. The thickness varies by color: 8-12 μm for black-dyed parts and 4-8 μm for clear parts. This process is ideal for achieving a smoother surface, offering good corrosion resistance and some wear resistance.

Type III anodizing, known as "hardcoat" anodizing, creates coatings up to 125 μm thick, with a standard thickness of 50 μm. This process forms dense layers that offer superior corrosion and wear resistance, ideal for functional uses. It requires more precise control than Type II anodizing, involving higher current density and maintaining the solution temperature near 0 degrees Celsius, which increases the cost.

Purpose: to impart a shiny, aesthetic finish that improves corrosion resistance
Compatible Materials: aluminum, titanium

Options

INFORMATION@HPPI.COM | 303-530-3131

POST-PROCESSING

Surface Finishing Methods

Galvanizing

Galvanizing involves dipping steel into molten zinc. This process coats the steel with layers of zinc-iron alloy and zinc metal. The zinc reacts with the steel's iron content, forming a uniform, thick protective layer.

Purpose: to improve the part’s corrosion resistance
Compatible Materials: mild steels, carbon steels, iron

Options

HIRSH PRECISION
PROTOTYPE PRODUCTION

Cutting Fluids

Cutting fluid is a specially designed substance used in metalworking and machining to act as both a lubricant and coolant. It is typically applied during machining through methods like flooding, fluid jets, or mist spraying.

A good cutting fluid will showcase the following properties:

Attribute	Description
Thermal Conductivity	This indicates how well it removes heat from both the workpiece and the tool.
Heat Capacity	A cutting fluid with a higher heat capacity can absorb more heat before it becomes hot or boils.
Low Viscosity	The low viscosity of a cutting fluid ensures smooth flow and prevents sludge from forming when chips mix with the fluid.
Non-Corrosive	Your cutting fluid cannot corrode or otherwise damage either your workpiece or cutting tool.
Corrosion-Oxidation Resistant	A quality cutting fluid should not only avoid being corrosive but also protect parts from corrosion and oxidation.
Non-Toxic	Given that your cutting fluid will come into contact with humans, choose a cutting fluid or coolant that isn't toxic.
Chemically Non-Reactive	Cutting fluids should not react chemically with surfaces they touch, as this can harm the surfaces and degrade the fluid.
Odorless	It should be odorless to prevent any lingering smells on the machine or the finished part.
Transparent/Translucent	It should be clear enough to allow an unobstructed view of the workpiece.
Stable	A good cutting fluid should remain stable and not break down quickly during use or storage.

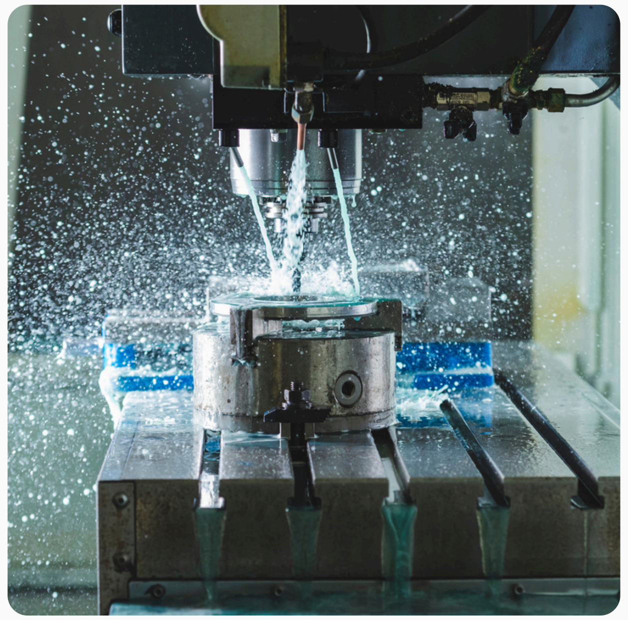
It's important to understand what types of cutting fluids are available on the market, the different properties and factors that will impact your machining project, and how to select a cutting fluid that will reduce costs, reduce impact on tools, and best protect your workpiece.

INFORMATION@HPPI.COM | 303-530-3131

CUTTING FLUIDS

Purpose

Cutting fluids and coolants are essential for improving the efficiency and quality of machining.



Lubrication

Friction is the main contributor of heat buildup during machining, which can cause surfaces to stick together. Cutting fluids help by forming a thin layer between the chip and the tool, reducing their contact. This lubrication also minimizes tool wear and decreases the energy needed for machining.

Improved Tool Life

Cutting fluids extend tool life by cooling them during machining operations, which minimizes the risk of overheating. By significantly reducing friction, these fluids ensure smoother and more efficient tool performance.

Chip Evacuation

During operations like milling and drilling, chips can gather around the cutting area, potentially hindering the operation. Cutting fluids help by removing these chips from the cutting zone.

Cooling

During metal cutting, heat builds up in the workpiece, chips, and cutting tool due to friction and metal deformation. This heat can cause problems like thermal expansion, oxidation, and surface welding. Cutting fluid helps cool the tool and workpiece, preventing these issues.

Corrosion Prevention

Cutting fluids contain rust and corrosion inhibitors that protect machine parts and surfaces from corrosion. Mineral oil-based cutting fluids prevent oxidation by creating a thin protective layer on exposed surfaces.

Improved Surface Finish

Cutting fluids help achieve a smooth surface finish on machined parts by preventing thermal expansion and changes in the workpiece properties.

CUTTING FLUIDS

Types

Cutting fluids are categorized by their phase, composition, source, and application method. They can be generally grouped as follows.

Straight Oils

Straight oils are mineral oils that do not mix with water. Initially, animal and vegetable oils were used as pure lubricants in metal cutting. While they are eco-friendly due to their biodegradability, they are costly and break down quickly. Therefore, they are now primarily used as additives to enhance the lubrication of petroleum and mineral oils.

Mineral oils are made from refining crude oil and are petroleum-based. They often include additives like chlorine, phosphorus, and sulfur to help reduce tool wear.

Soluble Oils

Soluble oils are mixtures made by adding mineral oil to water, typically in a ratio of 1-20% oil to water. These coolants include emulsifiers like sodium sulfate to help oil blend with water, additives to enhance corrosion resistance or act as coupling agents, biocides to stop bacteria growth, and anti-wear additives to improve lubrication.

Soluble oils are the most affordable and widely used cutting fluids in machining. They offer effective cooling and moderate lubrication, making them ideal for light cutting tasks.

Synthetic Fluids

These water-based fluids compete with mineral and petroleum-based oils. They are created by dissolving organic and inorganic chemical compounds in water, along with additives. These additives include lubricants, rust inhibitors, and corrosion inhibitors, which enhance properties like lubrication that might otherwise be reduced by the water content.

Semi-Synthetic Fluids

Semi-synthetic fluids blend synthetic fluids, water-based fluids, and soluble oil emulsions. They consist of 5-50% mineral oil, additives, and chemicals that dissolve in water to create tiny microemulsions. These fluids offer the benefits of both soluble oil and synthetic fluids.

INFORMATION@HPPI.COM | 303-530-3131

CUTTING FLUIDS

How to Choose the Right Cutting Fluid

Choosing a cutting fluid depends on factors like the type of cutting tool, the material of the workpiece, and the machining operation.

Workpiece Material

Metals are the primary materials that need cutting fluids. Here are some commonly machined metals and their recommended cutting fluids.

- Steel – straight oils with added lubricants
- Alloy steels – mineral oils
- Aluminum – soluble oils or mineral oils without active sulfur (active sulfur stains aluminum)
- Copper – soluble oils
- Stainless steels – straight oils containing excellent extreme-pressure additives
- Cast iron – None

Cutting Tool Type

Carbide tools can become extremely hot, making them prone to thermal shock, where parts of the tool expand unevenly. For these tools, use a synthetic cutting fluid with excellent cooling properties.

High-speed steels heat up, but not as much as carbide tools. Soluble oils and semi-synthetic fluids work well for them.

INFORMATION@HPPI.COM | 303-530-3131

CUTTING FLUIDS

How to Choose the Right Cutting Fluid

Choosing a cutting fluid depends on factors like the type of cutting tool, the material of the workpiece, and the machining operation.

Machining Operation

Machining operations like turning, milling, forming, and drilling occur at high speeds, requiring significant cooling. These processes need only moderate lubrication and pressure resistance, making synthetic fluids the best choice. Soluble oils can also be used.

For challenging machining tasks like broaching and thread cutting, effective lubrication is essential. These processes occur at low speeds and high pressures, so cutting fluids with superior lubrication and extreme pressure capabilities are necessary. Mineral oils with extreme pressure additives are the ideal choice.

Priority

In addition to these other factors, you'll also want to consider your goal and end-use. Use the table below to identify what the most appropriate metal cutting fluid will be based on the project's priority.

Precision Machining	Water-soluble fluids
Heavy-duty Machining (steel, titanium)	Straight oils or emulsions
High-speed Machining	Synthetic or semi-synthetic fluids

Factors

Workpiece Material

Cutting Tool Type

Machining Operation

Priority

HIRSH PRECISION
PROTOTYPE PRODUCTION

INFORMATION@HPPI.COM | 303-530-3131

CUTTING FLUIDS

Maintenance and Disposal

Proper cutting fluid maintenance extends tool life, prevents contamination, and cuts costs by ensuring top performance. Equally important is responsible disposal, making it crucial to follow best practices in fluid management and waste reduction.

Maintenance

Essential steps for maintaining cutting fluid include:

- Accurately monitor concentration levels (typically done with refractometers or chemical titration)
- Keep the pH level between 8.5 and 9.5 to stop bacteria from growing and keep the fluid stable
- Regularly remove tramp oil to maintain fluid efficiency and lower disposal expenses
- Install filtration systems to remove metal particles and contaminants, which extends the fluid's lifespan
- Provide thorough training so operators can recognize fluid degradation and carry out routine maintenance efficiently

Disposal

To reduce environmental impact and avoid legal issues, follow these practices to comply with environmental regulations:

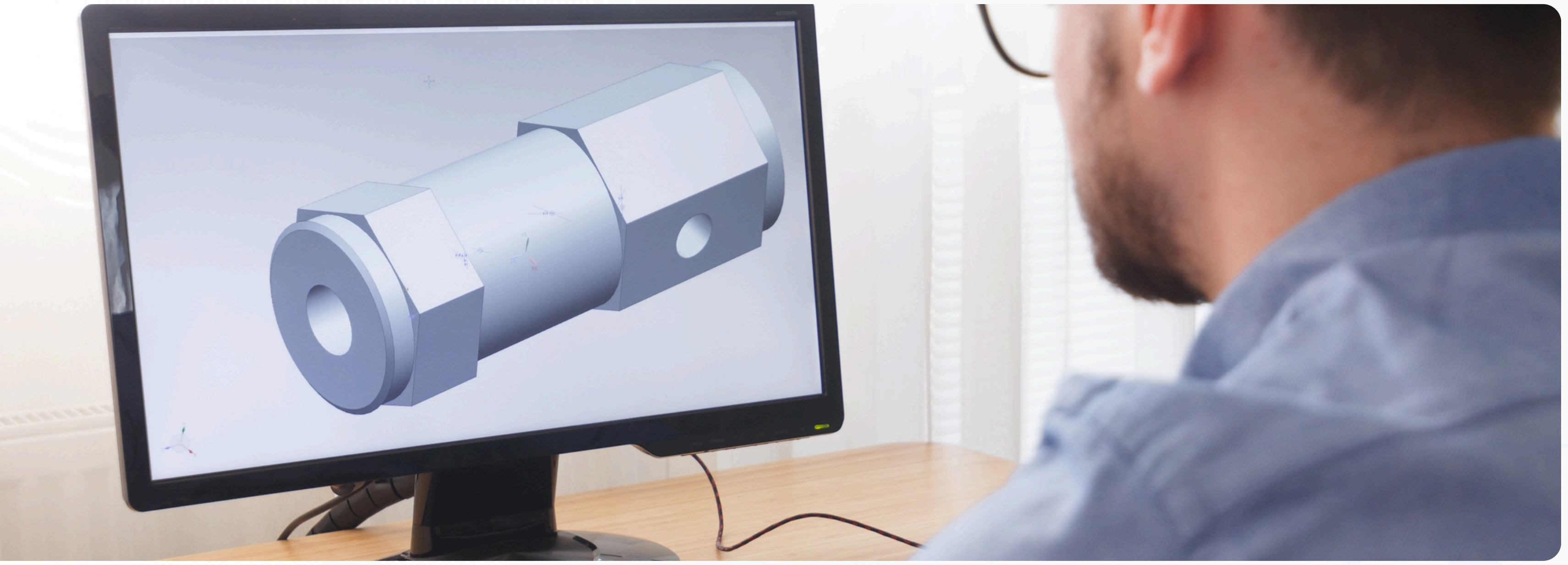
- Implement recycling programs that prolong fluid life by filtering and reprocessing coolants, which reduces both waste and costs
- Test coolants for harmful chemicals before disposal (done according to hazardous waste guidelines)
- Invest in coolant management systems that keep fluids in good condition and reduce waste
- Use biodegradable cutting fluids to reduce environmental impact

Machining Cost

In CNC Machining, costs vary for each project. Things like design, equipment, materials, and inspection needs all factor into the price tag.

Countless factors influence machining cost, but the main culprits are the following:

- Part design
- Material
- NRE and programming
- Raw material
- Machining process
- Tooling
- Post-processing
- Volume



MACHINING COST

Design

The design and shape of a part greatly influence the cost of CNC machining.

The more complex a part is, the higher the manufacturing cost. Complex parts often need advanced machinery, more machining time, multiple setups, additional resources, and thorough inspections -- all of which increase costs.

Certain design features can also increase costs. These include:

- Sharp internal corners
- Thin walls
- Deep cavities
- Non-standard hole sizes
- Lettering

Such features should be avoided unless necessary. Additionally, design specifications like surface roughness or tolerancing may need multiple passes and inspections, further raising costs.

Large parts cost more to produce because they need more raw materials, resources, and manufacturing time and effort.

INFORMATION@HPPI.COM | 303-530-3131

MACHINING COST

Fixed Cost Factors

Fixed costs represent the foundational expenses incurred at the start of a CNC machining project; these costs primarily include setup and programming. Regardless of the quantity of parts ordered, these costs are amortized across the entire production run.

This means that as the number of machined parts increases, the fixed cost per unit decreases, offering greater cost efficiency and value for larger orders. Understanding this distribution can help businesses optimize their budgets and make informed decisions about production volumes.

Programming and NRE

Programming and non-recurring engineering activities for machining jobs incur a significant one-time cost. For complex prototype parts, this can make up a large part of the total job cost. Further, the more complex features lead to more complex programming. Aim for the simplest machining strategy and workholding setups that use standard tools whenever possible.

Setup

Job setup is the process of preparing the machine and the workpiece for the machining operation, and is a fixed cost spread over the total number of parts produced. To lower these costs, follow DFM guidance similar to reducing programming expenses. Designing for simple or standard setups helps cut job setup costs in both prototypes and production.

MACHINING COST

Scaled Cost Factors

Unlike their fixed counterparts, scaled cost factors correspond directly to order volume. If you place a high volume order, you'll naturally spend more on these things compared to a low volume run.

Material

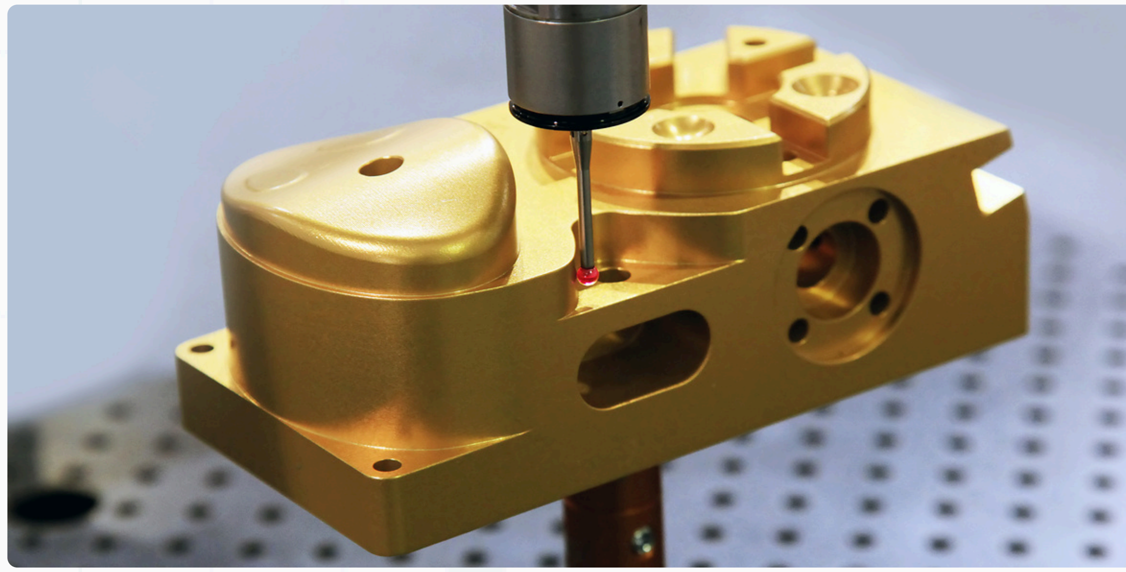
The material used to make a part is a key factor in its cost. There are various costs linked to machining a material, with the primary one being the material's own cost. Material prices vary based on their availability, desirable properties, and production costs.

Besides material costs, machinability significantly affects expenses. Materials that are difficult to machine require more time and effort, and in CNC machining, time equates to money. The longer it takes to machine a part, the higher the cost. Additionally, challenging materials use more resources, such as cutting fluids, electricity, and tools.

Machining Process

The chosen machining process is another key cost driver that you should keep in mind when approaching a CNC machining project. Costs related to a machining process include:

- Machine type and capabilities (i.e., 3-axis mill vs. 5-axis mill-turn machine)
- Machine time (i.e., spindle time, setup time, changeover time)
- Labor costs (i.e., required programming, setup, and operator costs; location has an impact here, as well)



Related Read: What to Consider when Calculating Material Cost

MACHINING COST

Scaled Cost Factors

Unlike their fixed counterparts, scaled cost factors correspond directly to order volume. If you place a high volume order, you'll naturally spend more on these things compared to a low volume run.

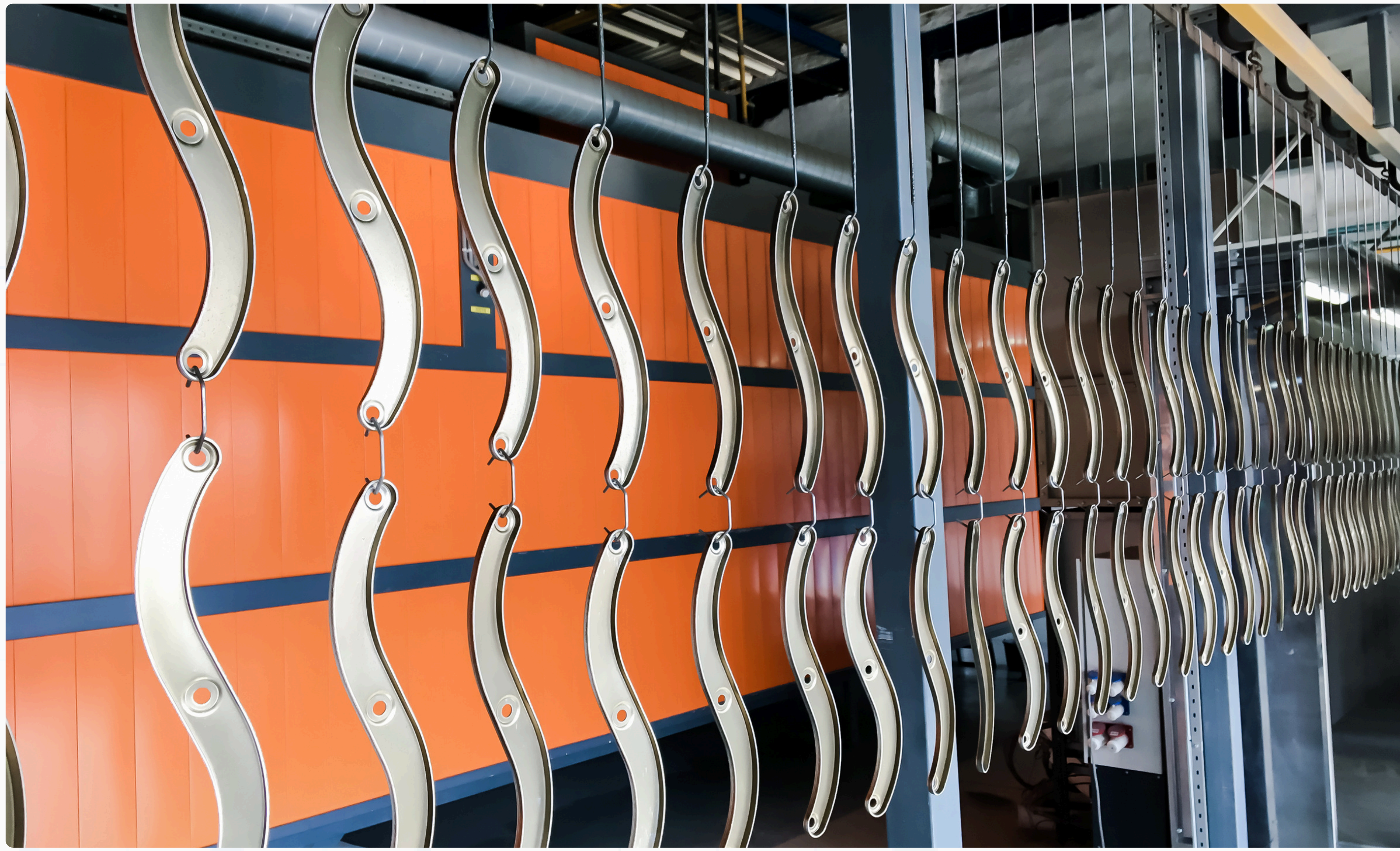
Tooling

Cutting tools also influence total equipment cost and scale accordingly with the volume of your order. Material, coating, and geometry all affect tooling cost, and costs are further extrapolated based on tool wear and/or breakage. For example, cemented carbide tools, which are harder, more heat resistant, and able to withstand extreme speeds, cost over two times more than tool steel tools.

Post-Proceses

Some applications need additional steps like heat treatment, surface finishing, and coating to enhance their performance, features, and appearance after the machining operation. And every component will require inspection. These extra processes add to the part's cost, and obviously those costs correspond to order volume.

Related Read: How to Build a Streamlined Tool Kit



MACHINING COST

Scaled Cost Factors

Taking part design, as well as fixed and scaled factors. into account, keep in mind that there is an inverse relationship between per-unit cost and production volume, often referred to as economies of scale.

While overall costs may be lower for low-volume production, they will have a higher per-unit cost. Concurrently, there are more upfront costs associated with high-volume production, but the per-unit cost is much lower.

Let's take a closer look at the relationship between production volume and cost.

Low Volume Production
Low-volume production is an essential and often necessary phase in the manufacturing process and product lifecycle.

Low-volume production requires minimal initial investment and is ideal for testing a product's viability before moving to large-scale production. However, optimizing these processes can be challenging since the cost benefits of mass production are not yet applicable.

Small order quantities often come with higher costs, and many CNC machining suppliers have minimum order requirements that might be more than you need for low-volume production.

High Volume Production
As you increase the number of identical parts produced, the cost per unit drops significantly. This cost reduction happens because you only need to handle the CAD design, CAM preparation, and machine set-up once for all parts.

Generally, ordering in larger quantities leads to cost savings throughout the supply chain. This includes savings on raw materials, castings, machined parts, post-processing, and more.

Resources

About Hirsh Precision

Hirsh Precision is a high-mix, high-volume CNC machining solution provider for the medical device, aerospace and industrial sectors.

Our DFM, machining, and post-machining processes support OEMs from the design stage all the way through product launch.

Resources

Website: hppi.com

Phone: 303-530-3131

Quote: quote@hppi.com

HIRSH PRECISION
PROTOTYPE PRODUCTION