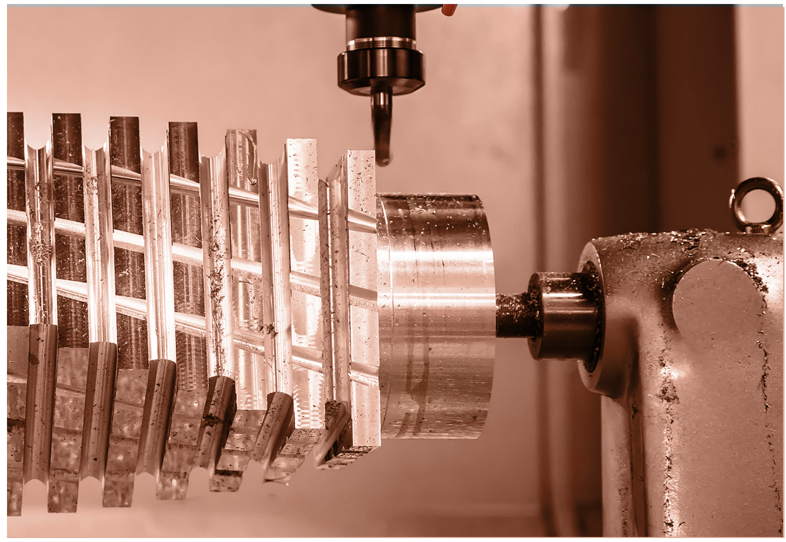
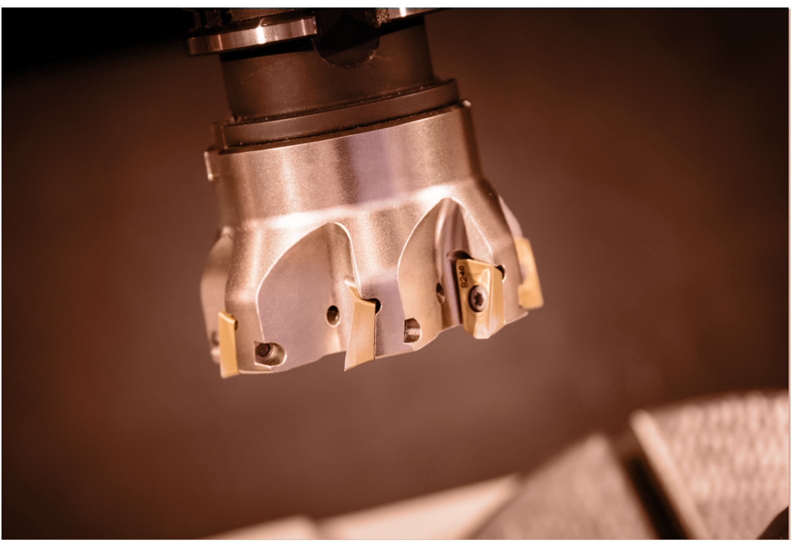
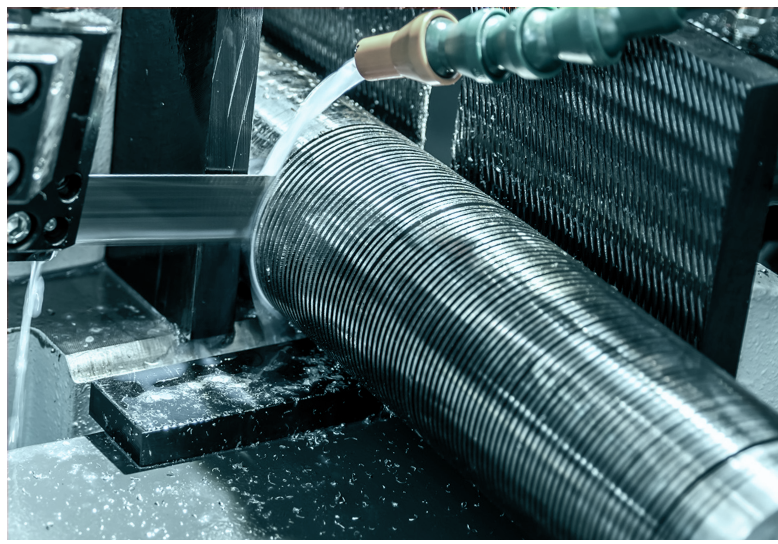
CNC MACHINING PROCESS GUIDE

HISTORY, CURRENT PRACTICE AND PROCESSES OF CNC MACHINING AND ADVANCED METROLOGY







Prepared by

Eagle CNC Technologies, Inc.
CNC Machining Facility
Muskegon, MI

www.eaglecnc.com

Table of Contents

Contact Us

Eagle CNC Technologies, Inc.
2357 Whitehall Road
North Muskegon, MI 49445

231-766-3914

www.eaglecnc.com

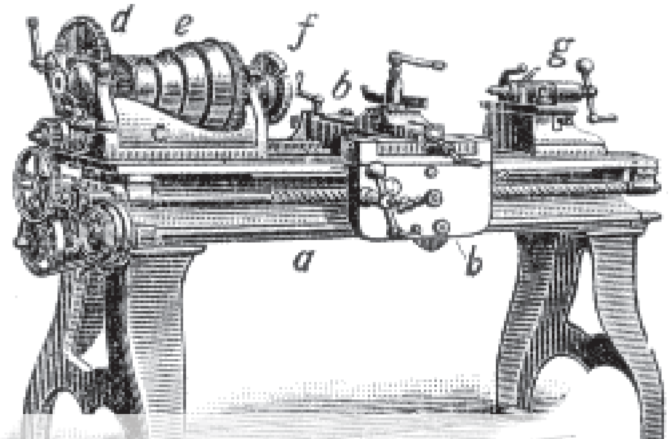
Introduction to CNC Machining

CNC Machining is often the last step in metal manufacturing, or sometimes the only process involved. Compared to other metalworking techniques, CNC machining is capable of meeting the tightest tolerances, and producing the most accurate, precise products with a high degree of repeatability.

To understand the origins of CNC Machining, it's important to break the phrase down into three parts:

- First, "**machining**," or the process of removing metal with the assistance of mechanical equipment, has been around for centuries. Machining processes include: turning, drilling, milling, sawing, broaching and grinding.
- Next, "**NC**" stands for "Numerical Control." In traditional machining, a human operator must control the motion of a machine tool. Numerical control, first developed in the 1940s, governs the motion of machines automatically, through set instructions.
- Finally, the "**C**" gets added. "Computer Numerical Control" designates a process by which an operator can write, adjust and implement instructions using a computer console. This addition, which took place in the 1950s, is what makes modern machining possible.

History of CNC Machining



Mechanical lathe, predating numerical control technolgy

In the late 1940s, John Parsons of Parsons Corp. in Traverse City, MI, developed a system to control machining equipment by feeding it punched cards with holes corresponding to coordinates. Starting in 1949, Parsons joined the U.S. Air Force at an MIT laboratory to further develop what was to become **numerical control**.

The team behind numerical control developed a process to produce aircraft parts that required a high degree of accuracy and precision, and took manual operators hours to work through. With the addition of numerical control, part after part could be produced with little human oversight.

The Advent of CNC

While early NC systems were revolutionary in their own right, there was still plenty of room for improvement. For starters, machine tools had to be programmed using a tedious process, and errors were common. Different companies developed different programming languages, resulting in confusion and incompatibility that, at first, stymied the growth of NC machining.

The U.S. Air Force stepped in once again, funding research at MIT to develop a universal NC programming language, which was first unveiled in 1959. Versions of the programming language, called Automatically Programmed Tools (APT) are still in use today.

In 1959 when APT was developed, computers weren't quite mainstream, but the technology grew quickly enough. Early NC machines could be programmed and controlled by small computers attached to each machine.

These days, many CNC machine shops make use of **Distributed Numerical Control (DNC)**, which allows a programmer to control several CNC units from a central computer. Many CNC interfaces are built around user-friendly software, and require much less training to operate than their forebears.

The Future of CNC Machining

As global demand for CNC machined products grows and changes, a wide variety of innovations are currently taking place in the CNC machining industry.

- **Robotics -** more than just mills and lathes, CNC is increasingly being applied to industrial robotics. In some factories, machines not only handle tasks like cutting and welding, but also transportation and assembly.

- **3D Printing -** both a competitor to CNC machining and a complementary process, industrial 3D printing is also growing rapidly. While machining is a subtractive process–starting with solid stock and cutting away material to form a shape–3D printing is additive. In many cases, designers can 3D print a prototype, and use it as a pattern to configure CNC equipment. In other cases, products can be accurately and precisely 3D printed straight from design software.

- **At-Home CNC -** The market for DIY CNC kits and inexpensive CNC machinery has grown exponentially over the past few decades. While they don't offer anything new in terms of processes, these at-home CNC stations make product development easier, faster and cheaper than ever. With the financial burden of metalworking vastly decreased, there's no telling what inventive products might come out of a hobbyist's garage.

CNC Machining Processes

The following standard machining processes are among the most common techniques used by machine shops today. While many other processes are in use, these are the principle methods by which machine shops perform work on metal parts.

Turning: Rotating the workpiece to bring metal in contact with the cutting tool. Lathes are often used for turning.

Milling: Rotating a cutting tool to bring it into contact with a stationary workpiece. Milling machines are used for milling.

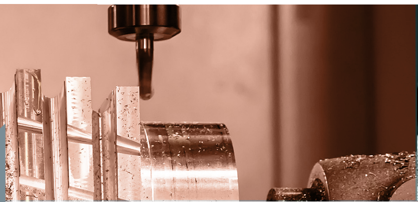
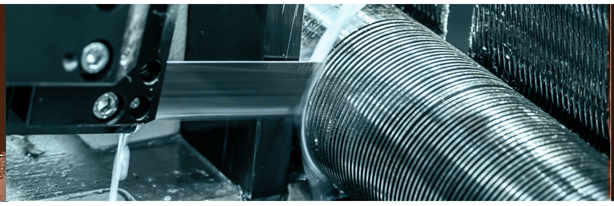
Drilling: Creating, or refining, holes by bringing a rotating cutting tool into contact with the workpiece. Milling machines or lathes are often used for drilling.

Boring: Removing material to form, or refine, highly accurate and precise inner cavities in a workpiece. Boring can be performed using a lathe or a milling machine, or a specialized boring machine.

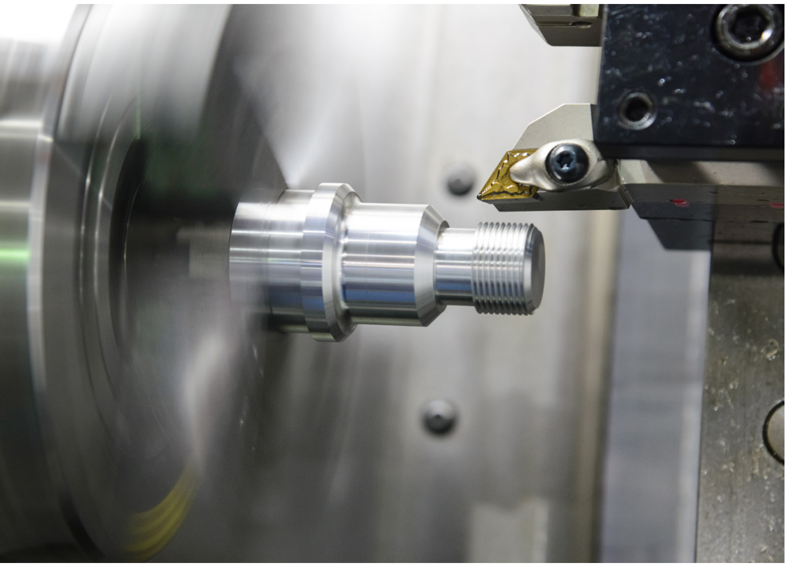
Sawing: Cutting a narrow slit in a workpiece, using a saw blade as a cutting tool. Saws or Sawing machines are used for sawing.

Broaching: Removing material through a series of shallow cuts, using a tool with many teeth of ascending height. The cutting tool is called a broach and is used in either a lathe or a broaching machine.

Grinding: Bringing a workpiece in contact with a rotating abrasive surface to smooth, or alter the shape of, a surface. Grinding machines are used for grinding.


Turning



Turning has been practiced by machinists for centuries on many different materials. Originally, turning acted on wood to create complex, cylindrical designs for use in tools, handles and furniture. Today, turning is a vital part of the metal manufacturing process, and a major technique used by precision CNC machine shops around the world.

Turning is the process of rotating a workpiece and bringing it into contact with a cutting tool. As the workpiece turns, the rotating motion forces the cutting tool to strip away material. The cutting tool itself can move linearly, either parallel or perpendicular to the axis of the spinning workpiece. Cuts made in turning, and the resulting shape of the workpiece, are determined by the motion and shape of the cutting tool.

What kinds of parts require turning?

Generally, turning produces cylindrical or conical surfaces. Because of the spinning motion involved, radial symmetry is often–but not always–a result of the process. **Thread cutting** is an example of a turning sub-process that does not result in radial symmetry. Bolts and threaded part sections are often machined this way.

Many parts and products depend on turning to create some or all of their surfaces. Some parts can be machined almost entirely through turning, like bolts, balusters and cylindrical tools. Other parts rely on turning to render their cylindrical or conical surfaces, and other processes, like milling or broaching, to create additional shapes.

What tools are used for turning?

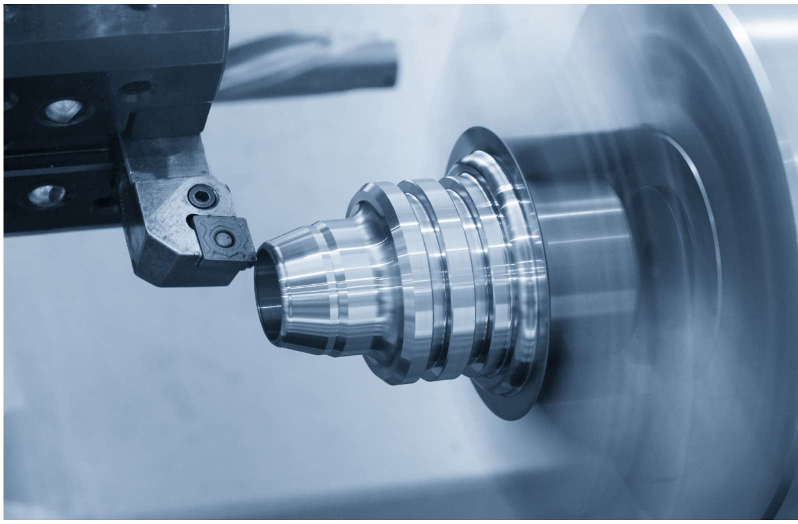
The **lathe** is the principal machine tool used for turning, but modern CNC machine shops have largely replaced traditional lathes with equipment that combines multiple operations, as in **mill/turn centers**, **machining centers** or **5-axis machining centers**. In addition to turning, lathes can also be used for other machining processes, like boring, reaming, facing or knurling.

Traditional lathes are comprised of the following parts:

- **Head stock** - contains a spindle that holds and rotates the workpiece
- **Tail stock** - positioned at the opposite end of the headstock and includes a "center," which is used to properly orient the workpiece along its axis
- **Carriage** - moves the cutting tool along the rotational axis, and holds the cutting tool and all mechanisms that govern the cutting tool's motion. In modern lathes and machining centers, the carriage often holds a turret, which allows tools to be switched mid-process.
- **Cutting Tool** - the part that comes into contact with the workpiece to remove material
- **CNC interface** - present only in Computer Numerical Control lathes, allows operators to program set parameters to govern the lathe's motion and speed

Mill/turn centers and machining centers often replace the carriage with a more versatile milling component, allowing the equipment to perform milling as well as turning. These milling components hold and move the cutting tools like a lathe carriage, but they allow greater articulation and also spin the tool at high RPMs to perform milling operations. Even without a dedicated milling apparatus, modern lathes in use today often provide basic milling capabilities, through turrets that allow for **live tooling**.

No matter how the equipment is built and configured, turning operations all function the same way: the machine spins the workpiece at a specified RPM, and the operator controls the selection and motion of the cutting tool.



Above and opposite: turning processes on a CNC lathe

Milling

Milling is one of the most common processes in CNC machining, most likely because it is so versatile. Using a single tool, machine shops can create contours, shapes and angles on the surface of a workpiece. Milling can completely transform a piece of metal stock into a finished part of nearly any complexity.

The milling process in CNC machining consists of removing material with a rotating cutting tool. Unlike turning, the workpiece does not need to rotate in milling operations. In some cases, the workpiece will move linearly against a cutting tool; in other cases, the workpiece will remain stationary while only the cutting tool moves.

Different Types of Milling

CNC Milling can be divided into several subtypes, based on the characteristic being milled, the specific tool being used, and the motion of the cutting tool along the surface of the workpiece. Milling subtypes include:

- **Face milling**: One of the most common milling operations, face milling involves a cutting tool with a rotational axis perpendicular to the surface of the workpiece. Face milling is often performed to create a precisely flat surface, or precisely shaped, shallow grooves.

- **Shoulder milling:** In shoulder milling, a spinning cutting tool travels along the edge of a workpiece, leaving an L-shaped "shoulder" of specific width. Often, shoulder milling is carried out to achieve a 90-degree angle between two outward-facing surfaces.

- **Profile milling:** When a spinning cutting tool cuts a path along a vertical or slanted surface, the process is called profile milling. Profile milling can be performed in a variety of ways, using a cutting tool that spins around an axis either perpendicular or parallel to the workpiece surface. Profile milling often involves movement around adjacent, non-planar surfaces.

- **Slot milling:** Slot milling forms a channel in the workpiece, often through the use of a disc-shaped cutting tool. However, slot milling can also be performed with a cutting tool that rotates along an axis perpendicular to the workpiece.

- **Chamfer milling:** Chamfer milling is performed with a certain cutting tool called a chamfer, in which the parts of the tool that engage the workpiece are positioned at an angle. The result is an angled surface, often replacing a 90-degree edge or surrounding a hole.

- **Thread milling:** One way to machine threads into a hole, or onto the outside of a surface, is thread milling. An alternative to tapping, thread milling uses a cutting tool that spins and moves along the surface being threaded. As opposed to tapping, thread milling can create threads on surfaces with a variety of diameters.

What tools are used for milling?

The milling machine, often referred to simply as a "**mill**," is the most basic tool used for milling. Milling machines vary widely, however, and many are combined into machining centers. In milling machines or in machining centers, the shape of the product is determined by the motion of either the workpiece or the cutting tool, along three linear dimensions (X,Y,Z) and up to three rotational dimensions (**pitch**, **yaw** and **roll**).

In the past, machine shops manually moved workpieces between lathes, mills, saws and drills to form a completed part. Today, versatile equipment can perform multiple processes on the same workpiece. Mill/turn centers, for example, combine milling capabilities with turning and boring. Many also include motorized turrets with **live tooling**, allowing the machine to combine rotary cutting with linear motion. CNC technology and 3D software help engineers choose the best strategy to produce precise, accurate machined products as efficiently as possible.

Whether milling takes place on a traditional milling machine or a 6-axis machining center, the process is the same: a rotating cutting tool is brought into contact with a workpiece to chip away material. The resulting form depends on the speed of rotation, the shape of the cutting tool, and the motion of the cutting tool along the surface of the workpiece.

Above: CNC face milling

Opposite: CNC slot milling

Drilling

Drilling is one of the most common techniques used in manufacturing to create holes. In contrast to other hole-making methods like boring, reaming and tapping, drilling is most often used to create holes in unbroken surfaces. In precision CNC machining, drilling can range in scope from simple, rough hole drilling to complex, multi-feature hole drilling.

Tools used for drilling

Many households have a common hand drill, used to make holes in walls or wooden surfaces. While this tool is easy to use and highly portable, it's not ideal for making accurate, repeatable holes in metal surfaces. A step in the right direction is the drill press. Also a common feature of woodshops, the drill press features a table that holds the workpiece steady, and a head that raises or lowers and holds a spinning drill bit.

CNC drilling machines can add various levels of complexity to the basic configuration of a drill press, but most function under the same principles. The following components make up nearly every CNC drilling machine:

- **Head**: responsible for holding the spindle and tool, and raising and lowering as the drill bit forms the hole.
- **Spindle**: the spinning shaft connecting the chuck and the head.
- **Chuck**: the component that grips the drill bit as it spins.
- **Platform**: while sometimes replaced with another component–in horizontal drilling machines, for example–the platform is responsible for holding the workpiece steady as the drill creates holes.
- **Drill bit**: finally, the actual tool doing the cutting is the drill bit. This component can come in a wide variety of shapes according to the size, texture and other characteristics of the hole being drilled.

The actual drill bit can make a big difference in the complexity of holes drilled. For example, **stepped holes**, or holes with multiple diameters, can be made in two ways. Using simple drill bits, an operator can start with a larger drill bit and drill to the desired depth for that diameter, then use a smaller drill bit and drill a smaller-diameter hole deeper into the workpiece. Another way to drill a stepped hole is to use a stepped drill bit, with built-in cutting surfaces corresponding to multiple diameters. Similarly complex drill bits can also be used to create a chamfer, or an angled surface, at the top of the hole.

When drilling is used in CNC machining

CNC machining facilities have a number of choices when it comes to making holes in workpieces, including boring, counter sinking, tapping and reaming. Drilling is used in specific situations: when it is either the best process for the job, or the most economical. Because simpler, less expensive tools can be used for drilling, the process is often employed during roughing stages. That is, an initial hole is created through drilling, and that hole will be adjusted to closer tolerances later with a different process.

Drilling is also ideal for deep holes. According to Eric Fazakerley, Process Engineer at Eagle CNC, deep holes are becoming more common as customers require these characteristics to support advanced mechanical operations. “When I first started, everything with drilling was just 3-5 times the diameter of the hole deep,” says Fazakerley. “We didn’t carry anything longer. Now we regularly drill holes up to 30 times their diameters in depth.”

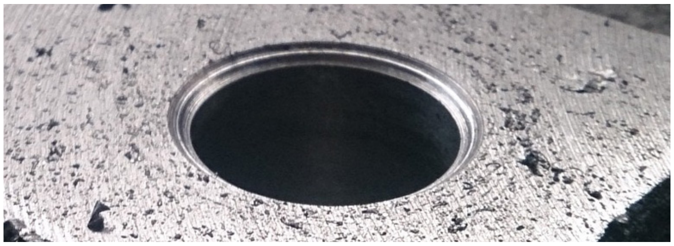
CNC drilling processes

The process of drilling doesn’t always involve a spinning drill bit and a machine head that moves up and down along the hole’s axis. For certain types of materials and certain dimensions of holes, the basic drilling process must be modified.

- **Spot drilling**: drilling shallow holes to hold longer drilling or boring tools in place during future drilling operations.

- **Peck drilling**: repeatedly plunging a drill bit into a hole and removing it along with metal ribbons, called swarf, created during the drilling process. Often used in relatively deep holes, where accumulation of swarf is more challenging.

- **Orbital drilling**: some drill bits are designed for orbital drilling, where the hole created is larger in diameter than the bit. To create a hole using this method, the drill bit “orbits” around the center of the hole. This process is similar to boring.



Above: a stepped hole formed by drilling

Opposite: CNC drilling with liquid coolant

Boring

Boring may not sound like a very exciting topic, but don't let names deceive you: boring is one of the most widely used techniques in machining, and one of the most reliable ways to finish holes.

Boring is the process of enlarging and finishing pre-existing holes. The holes might have been cast, drilled or otherwise formed to a rough state, but boring is often the best technique to provide the accuracy and repeatability expected of CNC machined parts.

Types of CNC Boring

Boring can be broken up into several subtypes, designated by the tools used and the types of hole being bored. For example, through-hole boring in which the workpiece is supported at both ends of the hole often falls under lineboring. Boring can also finish blind holes, but the workpiece can only be supported on one end. In backboring, the cutting tool traverses a through hole and works on the side of the workpiece opposite the headstock.

Just like drilling, milling and turning, boring processes can vary according to the desired shape of the final hole. Multi-feature holes can be bored with the right tools, including stepped bores, chamfers and counter bores.

Tools Used for Boring

Boring can be performed using just about any rotating CNC machine, including milling centers and turning centers. Specific machine tools for boring are also available: horizontal boring machines support the workpiece on an axis perpendicular to cutting rotation; jig borers are used to located the precise center of a hole before removing significant amounts of material.

Arguably the most important difference between one boring job and another is the cutting tool used. Some operations can make use of dual-point cutting tools, where the cutting surfaces are often 180 degrees apart. Operations requiring closer tolerances often make use of single-point cutting tools. Either type of cutting tool can be used on milling centers, turning centers or dedicated boring machines.

Beyond cutting tools, the right setup for the job depends on the characteristics of the hole being formed and the capabilities of the machine shop. Especially challenging holes may require custom equipment. For example, especially deep holes can be difficult to bore due to tool deflection. Some holes, like the SR-99 Tunnel in Seattle, WA, require dedicated borers that move independently through the hole as they cut. Analogous situations in CNC boring are rare, however, due to the more manageable scale of most CNC operations.

Furthermore, since the characteristics produced through boring are on the inside of the workpiece rather than the outside, more sophisticated inspection is often required. Laser-based metrology facilitate the process, and air gages can provide accurate measurements of bored holes.

When CNC Machine Shops Use Boring

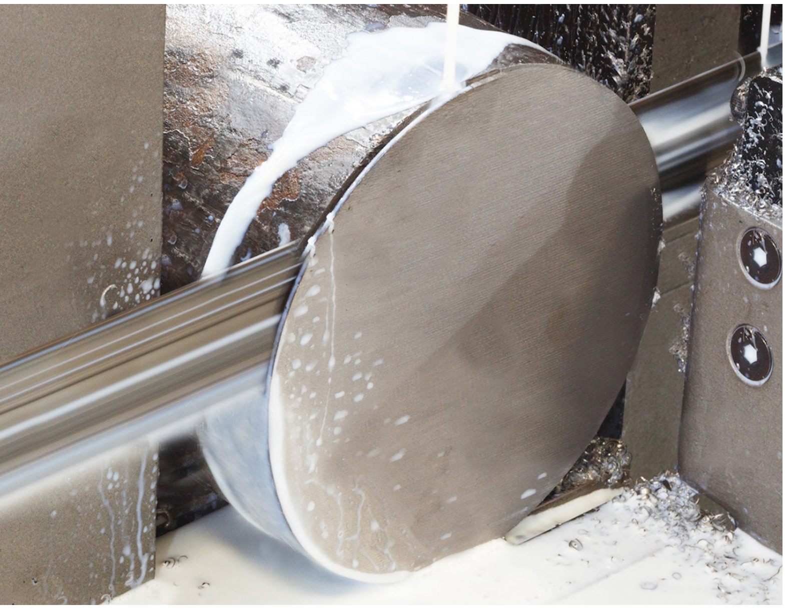
When choosing a process to create or enlarge a hole, CNC machine operators must evaluate a number of factors. Depending on the stage the hole is in and the desired characteristics of the final hole, operators might choose reaming or drilling instead of boring.

Drilling is often utilized to create a rough hole that will then be enlarged through boring and finished through reaming. If the operator uses the right tooling, boring can also be the finishing process.

Generally, boring is meant to reduce tolerances and improve precision in large runs of cast or machined products. The process is highly regarded for its accuracy, but it is not ideal for removing large quantities of material. If holes have not yet been started, a less accurate process like drilling is often used before boring. If medium tolerances are required, milling and turning can also be used to refine previously formed holes in a workpiece.

Above and opposite: holes created using the boring process

Sawing



Sawing is one of the oldest cutting techniques in use today, and innovations have allowed the process to keep up with advances in material, tolerances and product complexity. By definition, sawing is cutting a narrow slit in a workpiece by moving a toothed or abrasive cutting tool against the surface. Sawing is often used to remove large sections of material without particular concern for tolerances, but modern CNC sawing machines can be used for finishing work as well.

Types of sawing

At its most basic, sawing can be divided into two types: **continuous cutting** and **reciprocating cutting**. The reciprocating method involves saws that move back and forth and remove material on every stroke, or in many cases only on the forward or reverse stroke. Continuous cutting takes place when the saw moves continuously in one direction, constantly removing chips from the workpiece. Continuous cutting takes on different forms depending on the tools in use. For example, **bandsawing** employs a thin, flexible sawblade that runs in a continuous loop. **Cold sawing** involves a circular, toothed sawblade.

Continuous cutting also makes unique sawing types like **friction cutting** and **abrasive cutting** possible. In friction cutting, the high speeds of circular saws or bandsaws generate enough heat at a localized point on the workpiece to melt and carry away metal without chip formation. Blades for friction sawing often have no teeth. Abrasive cutting can be performed with a bandsaw or a circular saw, using a blade coated in an abrasive substance. The process then works similarly to grinding, where small chips are cut by the abrasive edges and carried out of the kerf by the rotation of the blade.

Tools Used for Sawing

Reciprocating Saw Machines:

- **Manual hacksaws:** the classic toolbox saw, with a handle and a thin blade held in a rectangular frame.
- **Power hacksaws:** mechanically

reciprocate a sawblade, similar to a manual hacksaw but much more rapidly and with greater force. Cutting usually only takes place on forward stroke.

Continuous Saw Machines:

- **Bandsaws:** can be horizontal, vertical or combination. Blades are formed from thin strips of metal that can be broken and re-welded to perform work on internal surfaces. Horizontal bandsaws are often used as replacements for power hacksaws, but bandsaws can offer greater efficiency. Vertical bandsaws can be used for contour cutting, as described below.

- **Contour saws:** a type of bandsaw used for cutting irregular shapes with a high degree of precision.

- **Circular saws:** traditional circular saws are "cold saws," or toothed disks spinning at high speeds. Other options include friction disks and abrasive disks.

When CNC machine shops use sawing

CNC machine shops often start with bar stock when machining a workpiece from start to finish. To get the bar stock down to the approximate size needed to produce the part, they saw a long piece of bar stock into smaller sections. Sawing's advantages all come into play here: the cutting action is quick, energy required is minimal, and very few chips are produced, meaning that little of the bar stock ends up in the scrap bin.

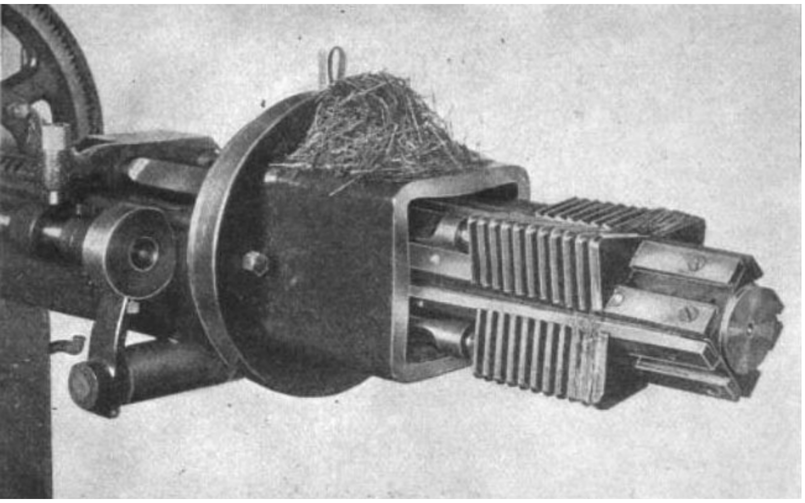
For simple cutoff operations like this, sawing usually uses less energy than other machining processes. Milling, turning and other processes could achieve the same goals, but they would either require highly customized cutting tools or would produce a far greater number of chips and would utilize much more energy. Because saws maintain a narrow kerf and can cut through just about any material given the right setup, they are ideally suited for cutoff operations.

Contour saws are equipped to cut irregular shapes with far greater precision than is needed for standard cutoff sawing. Contour saw machines can also offer comparable tolerances to milling machines, along with excellent repeatability. In many cases, contour sawing is used to cut tool steel in order to make dies for other machining operations.

Above: circular saw performing cutoff operation

Opposite: CNC horizontal bandsaw cutting disk from bar stock

Broaching



Broaching is a machining process using a cutting tool with teeth that increase in size from front to back. In many cases, an entire surface (or multiple surfaces) can be finished in a single pass with broaching. The technique is most often employed to finish holes, splines and flat surfaces.

How Broaching Works

The cutting tool used in broaching is called a "broach." This linear, toothed cutting tool resembles a saw in profile, but width and tooth configuration can vary significantly. Broach teeth are precisely designed so that each tooth stands slightly higher than the last. As the broach passes along the workpiece, each tooth makes a small cut in the surface and carries a chip away from the workpiece. The difference in height between the first tooth and the last tooth is called the "rise," and represents the maximum amount of material a broach can remove.

The process of broaching is relatively simple, with a broaching machine moving the broach linearly along the workpiece surface with sufficient force as to remove material.

Cutting Tools Used for Broaching

Broaches are often designed for specific jobs, and they can take on many shapes and sizes. There are several broach categories that relate to the work being performed and the broaching machine used to supply the force.

- **Push broach:** tool designed to be pushed by the broaching machine
- **Pull broach:** tool designed to be pulled by the broaching machine
- **Stationary broach:** tool designed to remain stationary while a machine moves the workpiece

Within these subtypes, broaches can vary in terms of tooth configuration and cross-section. The simplest broaches, designed to cut a single surface, may be rectangular in cross section with a single set of cutting teeth. Because broaches often cut internal characteristics, their cross sections vary extensively depending on the work being performed.

Various components of each broach serve specific purposes. The first few rows of teeth to come in contact with the workpiece are known as "chip breakers," while the last few rows of teeth are often "finishing teeth." Middle teeth are designated for various purposes, from roughing to semi-finishing.

Types of Broaching Machines

Simple broaching can be performed with an arbor press, but more complex operations require dedicated broaching machines. Broaching machine types include:

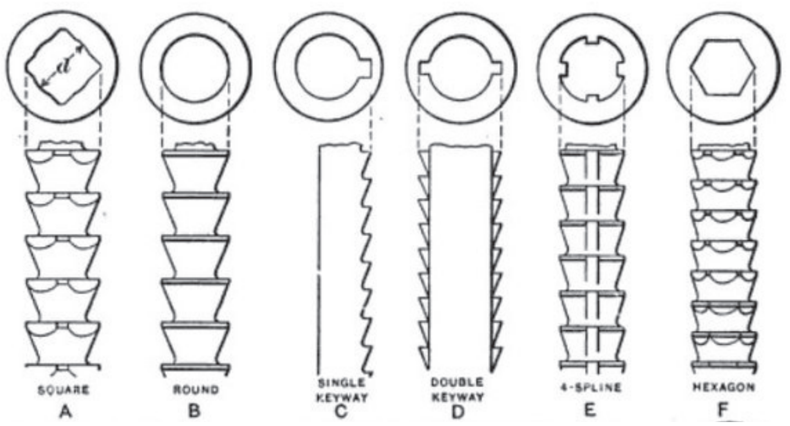
- **Vertical pull-down broach machines:** the pilot end of the broach is attached below the workpiece, with the chip breakers above the workpiece. The machine then pulls the broach downward.
- **Continuous surface-broaching machines:** broaches are stationary and the workpiece is pulled or pushed past the teeth in succession.
- **Rotary broaching machines:** workpieces are held in fixtures on a rotating table that passes them across stationary broaches. With the right configuration, rotary broaching can also be performed on a lathe.

Advantages of Broaching

One key advantage to broaching is that machine operators can perform broaching operations with little training. Because the complexity is built into the tool itself, very few parameters require adjusting during production. Perhaps most importantly, broaching can save time because the cutting process itself is rapid and easily repeatable.

Compared to sawing and grinding, however, broaching often acts on workpieces with greater force. Machining professionals must ensure that the workpiece material is strong enough to withstand the forces involved in broaching.

Ultimately, the choice to utilize broaching over other machining processes often comes down to three factors: volume, equipment and cut. In some cases standard broach designs can be used, but custom broaches are expensive to produce and so require high production volumes to be economically viable. Broaching machines are not as common in machine shops as mills, lathes and drills, so other processes may be used in order to avoid outsourcing or purchasing new equipment. Still, there are many cases when broaching is the only way to cut material in the desired pattern. Even with tooling adjustments milling machines, lathes, drills or bores cannot produce all the characteristics possible with broaching.



Above: possible profiles and cross sections of broaches

Opposite: illustration of horizontal broaching machine with broach attached

Grinding

Grinding is a machining process using abrasive surfaces to remove material from metal workpieces. On the surface (pun intended) grinding may seem different than other machining processes, but it still works through chip formation and removal–just like sawing, milling, broaching and most other techniques. Grinding can produce surfaces conforming to rough or extremely close tolerances. As such, grinding is used for simple gate removal in castings as well as advanced finishing processes like polishing and sharpening.

How Grinding Works

From a physical standpoint grinding, milling, turning and most other metal removal processes involve **compression and sheer**: a cutting point compresses the workpiece material until it breaks off (sheers) in the form of a chip. Sharp points on grinding wheels or belts sheer chips, which are then carried away from the workpiece in the cavities of the surface. Grinding's principle difference is that instead of using sophisticated, precisely formed cutting tools, it utilizes abrasive materials that supply the sharp points needed to sheer metal chips.

Tools Used for Grinding

Grinding can either be accomplished using a grinding wheel or a grinding belt. The composition of a grinding wheel depends on its application, but most are formed by combining abrasive materials with bonding agents and molding the mixture into the shape of a wheel.

Structure is a term referring to the spacing of the abrasive particles on a surface. Grinding surfaces with low density structure, or open structure, have larger chip cavities and so are able to remove material more quickly. However, open structure abrasives do not provide the same smooth surface finish as denser ones. **Grade** refers to the strength of the bonds that hold together abrasive particles in a grinding wheel.

In belt grinding, abrasive materials are affixed to a flexible backing cut into the shape of a belt. Belt grinding can be used for various applications, and works under the same principles as wheel grinding.

All grinding machines must be able to spin the grinding wheel at high speeds, or spin the wheels that carry the grinding belts, and must offer support to hold the workpiece. Rough grinding is often accomplished by holding the workpiece by hand, but precise operations require a more sophisticated motion-control system.

Types of Grinding

Some types of grinding in use today include:

- **Snagging**: a rough process using open-structure wheels, often used for removal of gates, risers and rough spots from castings. Snagging is used when accuracy is not a priority.

- **Creep-feed grinding**: a type of grinding that removes material quickly, at a rate comparable to milling. Modern versions require constant dressing of the grinding wheel.

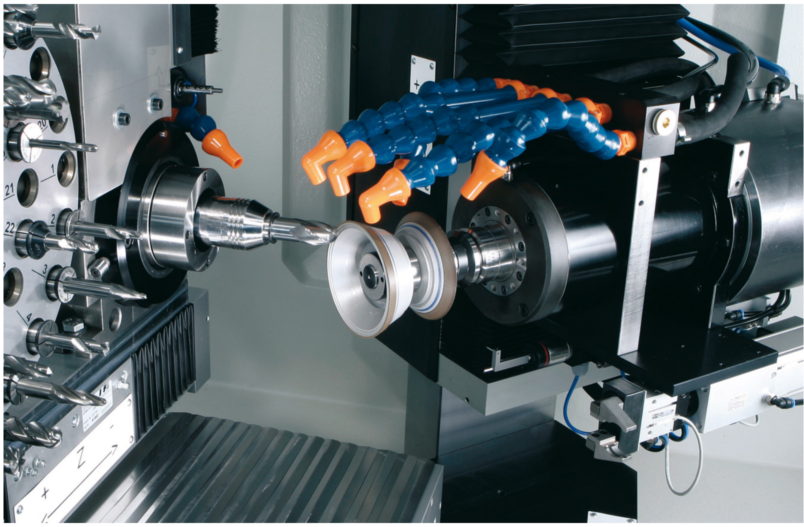
- **Cylindrical grinding**: a process used to grind cylindrical surfaces and contours of a workpiece. Cylindrical grinding is capable of finishing parts within extremely close tolerances.

When CNC Machine Shops Use Grinding

The grinding process is extremely versatile, and almost any size and shape of workpiece can be finished on modern grinding equipment. However, grinding is usually not as efficient at removing material as other machining processes, so it is often reserved for the final steps of manufacturing. Two exceptions are continuous-dress creep-feed grinding (**CDCF**) and **cut-off grinding**. Cut-off grinding actually mimics sawing, but uses an abrasive blade instead of a toothed one.

Sawing and grinding are often used together for gate removal on cast parts. Once a casting solidifies, the operator cuts away the gating with a saw. He or she then grinds the residual material on the casting to create a smooth surface.

Grinding is also commonly used in machine tool production. Tool steel is very hard and often cannot be cut by non-abrasive machine tools. **Superabrasives** can be used to form, sharpen and refurbish tools made from nearly any material, including carbide.



Above: a CNC grinding machine finishing tool-tips

Opposite: a CNC grinding machine finishing the surface of a part

Introduction to Metrology

Metrology is the science of measurement. In CNC machining, principles of metrology need to be applied throughout the production process to ensure that each part is machined to the right size and shape, in accordance with specified tolerances.

It's simple enough to take a ruler or a measuring tape and mark out inches or millimeters. But measuring dimensions of CNC machined parts – many with tolerances tighter than +/- .001 inch – is a whole other ball game. When you're dealing with measurements that small, you need to have sophisticated measuring tools, and the knowledge to use them.

Metrology and CNC Machining

CNC machining and metrology are closely linked, both in practice and in theory. Whether a machine shop is producing a fully machined part or finishing a cast part, their work is the end of the line. It's the last stop before the part is put to work, or integrated with a system of parts.

As applications for machined parts grow in sophistication, machining tolerances must also tighten to keep pace. Innovations in metal casting have allowed for tighter tolerances throughout the manufacturing process, but it's still up to CNC machine shops to be at the forefront of innovation in delivering the most accurate and precise parts possible.

Accuracy, Precision and Tolerance

- **Accuracy** refers to how close a measurement is to a standard value. An accurate part is one that is produced to dimensions at, or very close to, the intended standard measurements.

- **Precision** is accuracy through repetition. In CNC machining, precision means that every instance of a part will have the same, or very similar, dimensions. In metrology, precision can also mean that multiple readings give similar measurements.

- **Tolerance**, another term you'll hear quite often in the CNC machining world, is the degree to which a dimension is allowed to vary without making the part defective. This value is usually expressed as a range, as in "+/- .004 inches."

Measuring Standards

Modern manufacturing wouldn't be possible without rigorous standards of measurement, along with sophisticated tools and techniques to help us measure tools, workpieces and finished products.

CNC machining is a prime example of the practical uses of modern measuring, or metrology. In metal-based manufacturing, CNC machining is often the last process before a part is delivered. Without incredibly accurate and precise ways to measure and cut, many of the metal products we take for granted would be impossible to produce.

The history of measurement is the story of how human civilization strove to find new ways to measure accurately, and it all leads up to modern metrology.

Metrology all started with the idea of standard units of measurement. Around 6,000 BCE, civilizations had already developed measurement standards in order to calculate crop distribution and food consumption.

From then on, a wide variety of measuring standards have been used. Many were based on body parts, like the Egyptian 'cubit,' which measured the length from the elbow to the tip of the middle finger. Perhaps the most scientifically rigorous standard unit is the meter, which was first defined to equal one forty-millionth of the Earth's circumference, and was determined through an 8-year, multinational survey mission. Later, the meter was re-defined based on measurable wavelengths of light.

Measuring Standards Used Throughout History

- **The Egyptian cubit (circa 3,000 BCE)**: approx. 43-53cm, based on the distance between the elbow and the tip of the middle finger. Actual length varied depending on which Pharoah was in power.

- **The Roman Mile:** 1,000 two-step paces; actual length varied, but this distance is very similar to today's mile. 5,280 feet was declared the official measurement by Queen Elizabeth I in 1593.

- **The Metric System:** a universal measuring system first officially introduced in 1795. Spacial dimensions are based on the meter, which has had several different standards over the years. The meter is currently defined as the "length of the path travelled by light in vacuum during a time interval of 1/299 792 458 of a second."

- **Imperial Measuring System:** introduced in the United Kingdom in 1824; now still widely used in the United States, and includes yards, feet and inches. Inches have been based on barley grains, poppy seeds and a king's thumbnail.

Metrology Today

The sophistication of our measuring units coincides directly with our ability to measure more accurate dimensions. We've already come a long way from measuring the Pharaoh's forearm to measuring wavelengths of light, and recent decades have seen even more advancements in metrology.

As our measuring demands have become more strict, our measuring standards have also changed. We are no longer content to call an inch "three barley grains laid end to end," but now define it in reference to the metric system (an inch is officially 2.54 cm.) When manufacturing microscopically intricate products like microchips, or even threads on a machined screw, we need to be able to measure to an extremely high degree of accuracy, or the part will not function as intended.

How to Measure Accurately

CNC machine shops and other facilities relying heavily on accurate and precise measurements make use of CMMs, or Coordinate Measuring Machines. CMMs allow metrologists to measure not only three dimensions, but also shape-specific characteristics like **flatness**, **angularity** and **concentricity**. By using machines to speed up the process, they can take hundreds, even thousands, of individual measurements, and can create accurate computer models of parts. This allows experts to track quality markers, reverse engineer parts, and diagnose problems.

With CNC machining, quality means a high degree of accuracy and precision in every batch, every time. **When you work with a CNC machine shop, make sure they have a metrologist on staff and up-to-date equipment.** There's no way to guarantee quality if you don't have the tools and the skills to measure it.

Essential Metrology Tools

Every CNC machine shop will use a different set of metrology tools, depending on the type, quantity and quality of work they do. The variety of available tools go a long way in determining the overall capabilities of the facility.

Here's a list of metrology tools you can expect to find in modern machine shops:

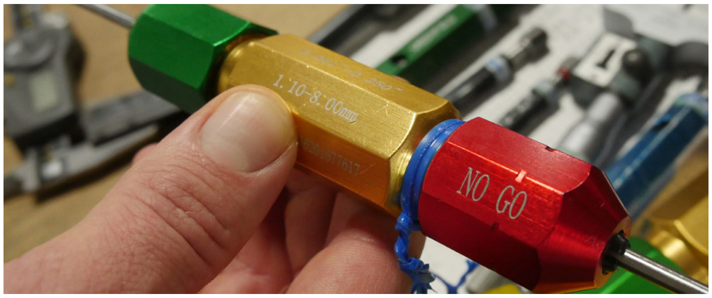
Surface plate

A surface plate is a flat surface used as a reference point for vertical (Y-axis) measurements. They're only useful if they're properly calibrated and maintain their shape impeccably. Because of its smooth surface and low level of expansion and contraction with temperature changes, granite is often the material of choice for surface plates.

Go/No-Go Gage

Go/No-Go gages measure parts to ensure that they are within their specified tolerances. The "no-go" portion represents one end of the tolerance range, and the "go" portion represents the other.

For example, in a plug gage, the "go" end should fit into the hole, while the "no-go" end should not.



Calipers

Calipers range broadly in shape, size and sophistication. Regardless of type, calipers all measure using the same principle: by allowing two opposing tips to rest at the beginning and end of a distance being measured. The common protractor, used to measure degrees, is also a type of caliper.

- **Vernier calipers:** first perfected in the 1600s, vernier calipers consist of a flat bar along which two opposing tips slide. The distance between the tips is measured using an accurately graded scale.

- **Dial calipers:** dial calipers function similarly to Vernier calipers, but display measurements on a simple dial, making them easier to read.

- **Digital calipers:** like dial calipers, digital calipers offer an easy-to-read display–in this case, a digital screen.

Micrometer

Micrometers use a calibrated screw, connected to a scale that moves as the screw is turned clockwise or counterclockwise. Many micrometers look and function similarly to calipers, but they can come in a variety of shapes.

- **Caliper micrometer:** general description of any micrometer made up of two opposing parts joined by a frame; often similar in appearance to Vernier calipers, but replacing the flat bar with a screw.

- **Bore micrometer:** a micrometer attached to a bore gage, used to measure the size and shape of holes.

- **Depth micrometer:** a micrometer used to measure the depth of any recess, hole or slot.

Air Gage

Air gages use pressurized air to measure the dimensions of an object. They were first introduced in the 1940s, and have undergone improvements to become one of the most trusted precision measuring tools in machine shops. Air gages function by shooting air out of a specifically calibrated nozzle, and recording the rate at which the air returns to multiple sensing nozzles. Thus, they are able to read the air flow between the measuring tool and the object being measured.

Coordinate Measuring Machine (CMM)

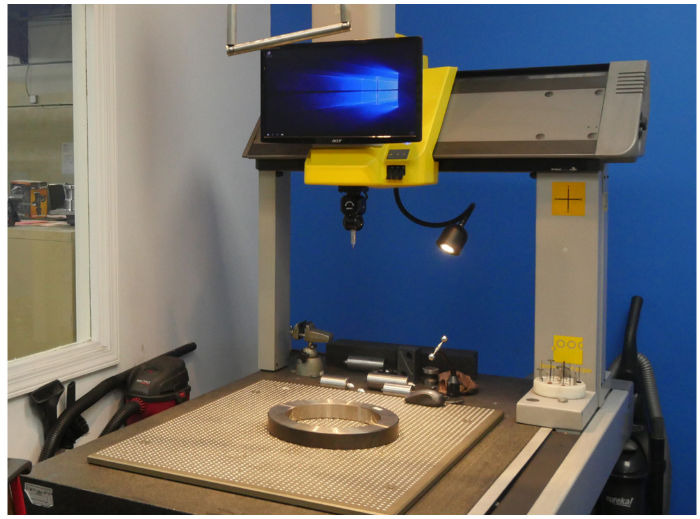
Dating back to the 1950s, CMMs are mechanical devices designed to automate various aspects of measurement. They are often shaped like an upside-down “U,” with legs that move on either side of a surface plate. The probe also moves side-to-side along the bridge of the “U,” as well as up and down, allowing measurement of X, Y and Z axes.

CMMs vary widely when it comes to the **probe**, or the attachment that directly measures an object through contact or proximity. Originally, all probes were mechanical, and recorded coordinates through direct contact with a surface. Optical probes, more recently developed, have a camera-like component that scans images at specified coordinates, registering changes in contrast within the image to determine accurate measurements.

Modern probes use lasers or white light to scan objects, recording thousands of data points at a time. This technology makes it possible to integrate with CAD software and produce virtual 3D models of parts with a high degree of accuracy.

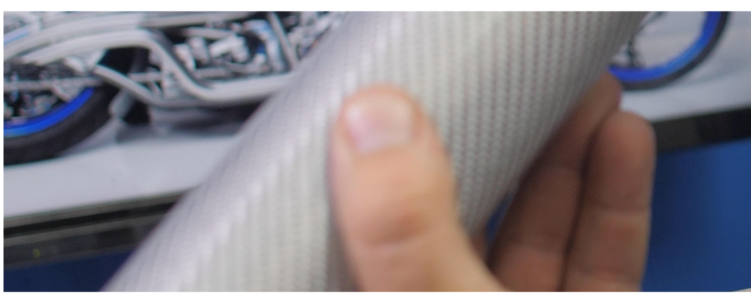
Laser Scanner

State-of-the-art machine shops can take metrology to the next level with laser scanners. These devices, a type of portable CMM, allow metrologists to measure a broad range of dimensions simultaneously, and are capable of making millions of calculations per second. Some laser scanners are handheld, like the one used by Eagle Alloy. Some are connected to articulated arms that allow for more rigidity, as well as controlled coordinate measurements. Both types are capable of measuring thousands of points at one time, creating a digital mesh of an object. Engineers can then compare that mesh with a 3D CAD file to determine where, and how much, each part deviates from nominal dimensions.



Above: CMM in standard configuration

Below: laser scanning arm at Eagle CNC



About Eagle CNC Technologies, Inc.

Eagle CNC Technologies is a Muskegon, MI-based company specializing in the CNC machining of both ferrous and non-ferrous castings, forgings, bar stock and burnouts. Whether you are looking for individual parts or finished assembly, we have the experience necessary to serve as your single-source provider.

Eagle CNC Technologies is one of four independent companies that make up **the Eagle Group**. Together, the Eagle Group provides full-service metalcasting and CNC machining services utilizing a wide variety of state-of-the-art tools and techniques.

To learn more, visit us on the web at **www.eaglecnc.com**

Contact Us

Eagle CNC Technologies, Inc.
2357 Whitehall Road
North Muskegon, MI 49445

231-766-3914

www.eaglecnc.com