

Lesson 1

Introduction & CNC Process Overview

Upon successful completion of this lesson, you will be able to:

- Describe overall course goal, prerequisites, audience and course design.
- Describe the difference between Subtractive Rapid Prototyping (SRP) and Rapid Prototyping (RP) and the advantages and disadvantages of each.
- List the tools and equipment required for this course.
- List the major lesson topics covered by this course.
- List the major student resources available and the information found in course appendices.
- Describe the recommended use of course materials to maximize the learning experience.
- Describe in general the overall CAD/CAM/CNC process.

1.1 Course Description

The goal of this course is to teach persons with a technical background how to program and operate Computer Numerical Control (CNC) mills and lathes.

This course bridges the gap between what persons with a technical education know and what they must know to begin using CNC machine tools. The types of parts, materials and machining operations that engineers, innovators, and niche manufacturers often use are featured. Work holding techniques well suited to prototype and short-run production are detailed and used in the examples.

Prerequisites

The learner is expected to have the following:

- Engineering or associates-level technical degree or equivalent experience.
- Knowledge in the proper use of basic hand tools and precision measuring instruments, including calipers and micrometers.
- Some manual machining experience is helpful but not required.

Audience

This course is designed for the following audiences:

- Working engineers and designers.
- Engineering undergraduates or graduate students.
- Manufacturing and design students.
- Technical school students.
- Scientists.
- Innovators.
- Niche manufacturers.

Course Design

CNC Machining is a very broad subject and there are many ways to do most things. Covering all options would fill volumes and is beyond the scope of any one book or course. The goal of this course is not to turn an engineer into a journeyman machinist. Rather, it is to show how to use CNC's to make relatively simple parts, teach DFM principles, and help engineers become better designers and managers.

This curriculum was created using an Instruction Design process. Engineering educators and students from leading Universities, as well as practicing engineers in a variety of industries, were surveyed. This process determined the types of parts and materials covered. Parts that are more easily made using Rapid Prototyping (RP) technologies were excluded.

By leveraging what anyone with a technical education knows, by focusing on the most common types of parts and materials, and by presenting best practices for prototype machining, learning objectives were considerably narrowed. Using this approach, a remarkable amount can be achieved in a short time. For example, working engineers using this course have been taught to set up, program, and operate a CNC mill in less than 8 hours of combined classroom/lab time; including instruction in HSMWorks/Inventor CAM software.

This economy of instruction makes CNC accessible to almost anyone: from working engineers to students involved in design/build competitions, to undergraduate engineering students as part of a Design for Manufacturing (DFM) course or hand-on lab.

SRP vs. RP

This course emphasizes an approach to CNC machining referred to as **Subtractive Rapid Prototyping (SRP)**. SRP deals with small quantities of functional prototypes. Functional prototypes are made from materials like aluminum, steel and polycarbonate that cannot be produced with widely available additive Rapid Prototyping (RP) processes such as SLA (Stereolithography) or FDM (Fused Deposition Modeling).

CNC machining is not as simple to learn and use as RP. It takes more skill and often more time to apply. The main advantage of CNC is in freedom of materials. Almost anything can be machined. Machined parts are precision structural components that can be tested and assembled as part of working machines.

Another advantage of machining is that it teaches real manufacturing constraints typical in the aerospace, biomedical, consumer goods, and electronics industries –all which use CNC for mass production, molds and other tooling. RP does not reflect these constraints. A part that is easy to rapid prototype may be extremely difficult, expensive, or even impossible to manufacture. SPR provides the designer with feedback about the manufacturability of a design that can save considerable time and money as a part moves from concept to product.

Prototype vs. Production Machining

A big difference between making a few or many of anything is the design of work-holding fixtures. Prototype machining emphasizes quick, simple and cheap work holding solutions such as vises, clamps, screws or even glue or double-sided tape. High production parts allow the cost of fixtures to be amortized over larger quantities that justify the cost of more elaborate and efficient fixtures. This course emphasized prototype fixturing; often quick, cheap and effective when making just a few of something.

1.2 Required Tools and Equipment

CNC Machine

To maximize learning, this course focuses on CNC machines made by Haas Automation, Inc. of Oxnard, CA. Chapters 5 (CNC Programming Language) and 6 (CNC Operation) are written specifically for the Haas control. Haas machines are highlighted for the following reasons:

- Haas Automation is the largest machine tool builder in the western world.
- Haas has donated or endowed many machines to colleges, universities, and technical schools, so they are likely to be found in educational institutes.
- All Haas machines use the same control, work similarly, and use industry standard programming. What is learned is easily transferred to other make or model machines.
- Haas makes several small footprint CNC machines designed specifically for engineering, prototyping, medical, jewelry, and niche manufacturing industries.

Warning: While this course covers the safety, setup and operation of the Haas CNC's, it is not a substitute for the machine operator manuals or training by a qualified technician.

HSMWorks/Inventor HSM

This course uses SolidWorks®, Autodesk® Inventor® and Fusion360® CAD software and HSM for CNC programming. SolidWorks, Inventor and Fusion360 are widely used by both industry and education for mechanical design. HSM is completely and seamlessly integrated into all three products, is very easy to learn and use and is particularly well-suited to persons who are new to CNC programming. HSM also stores all

data in the CAD Part or Assembly file which makes file management simpler. If the learner already knows how to use Inventor, SolidWorks and Fusion360, they already know most of what they need to use HSM.

HSM makes a streamlined course in CNC possible. It saves the substantial time it would take to learn a completely separate user interface and way of working. Our experience is that the distraction and time needed to learn “stand-alone” CAM is a major obstacle to learning CNC. Besides adding no value to the learning experience, the long slog it takes to learn another (and often quirky) CAM interface is discouraging to students who, like everyone else, just wants to start making something now.

HSM cuts CAM learning time easily by 75% or more. This frees class time, reduces learner frustration and focuses attention where it belongs: on how to set up, program and run CNC machine tools –rather than teaching mouse clicks of yet another software application.

1.3 Lessons & Appendices

Lessons

- 1 – Overview/Resources
- 2 – Shop Safety
- 3 – Coordinate Systems
- 4 – CNC Programming Language
- 5 – CNC Tools
- 6 – CNC Operation
- 7 – 2D Milling Toolpaths
- 8 – CNC Turning

Appendices

- A – CNC Milling Work-Holding Examples
- B – Alternate Tool Setting Methods
- C – CNC Process Planning Forms
- D – G/M Code Reference
- E – Glossary
- F – Drill Chart

1.4 Instructional Resources

Instructor Resources:

- PowerPoint presentations for each lesson.
- Lesson Guides.
- Exercises: SolidWorks files with HSMWorks toolpaths completed.
- Videos with step-by-step instructions how to complete each exercise.
- Assessment with solution for each lesson.

Student Resources

- Reading assignments for each lesson.
- Videos (SWF) for each lecture (PowerPoint).
- Videos with step-by-step instructions how to complete each exercise.
- Exercises: SolidWorks files.

1.5 Recommended Use

To get the most from this course, proceed as follows:

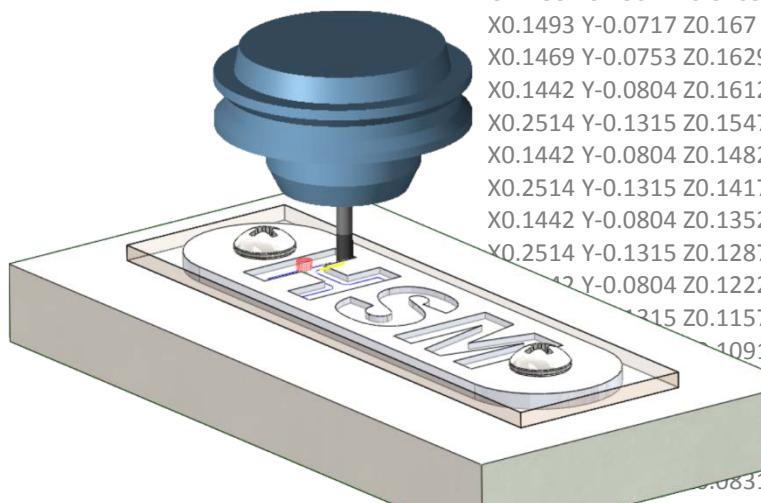
- Read the Reading Assignment for each lesson (PDF).
- Watch the video of each lecture, or attend a live lecture conducted by the instructor.
- Watch the video for the practical assignment.
- Complete the assignment using HSMWorks. Work through the assignment more than once if possible. Make sure you understand each step and why you are doing what the assignment requires.
- Make the project parts on the CNC machine.
- Take the assessment for the lesson.

1.6 Overview of CAD/CAM Process

The general workflow to go from CAD model to machined CNC part is:

1. Begin with CAD model.
2. Establish Job parameters including CNC coordinate system and stock shape/size.
3. Select CNC process.
4. Select cutting tool and machining parameters.
5. Select driving CAD geometry.
6. Verify toolpath.
7. Post Process.
8. Transfer G-code program to CNC machine.
9. Set up and operate CNC machine to make part.

```
%  
O01001 (CNCFUN1.6-1 NAMEPLATE EXERCISE)  
(T1 D=0.125 CR=0. - ZMIN=0.075 - flat end mill)  
G90 G94 G17  
G20  
N1 T1 M6  
S5000 M3  
G54  
G0 X0.1504 Y-0.0701  
G43 Z1.434 H1  
M8  
Z0.1906  
G1 Z0.1737 F13.333  
G17 G3 X0.1501 Y-0.0705 Z0.1702 I0.0473 J-0.0359  
X0.1493 Y-0.0717 Z0.167 I0.0476 J-0.0355  
X0.1469 Y-0.0753 Z0.1629 I0.0485 J-0.0343  
X0.1442 Y-0.0804 Z0.1612 I0.0509 J-0.0306  
X0.2514 Y-0.1315 Z0.1547 I0.0536 J-0.0256  
X0.1442 Y-0.0804 Z0.1482 I-0.0536 J0.0256  
X0.2514 Y-0.1315 Z0.1417 I0.0536 J-0.0256  
X0.1442 Y-0.0804 Z0.1352 I-0.0536 J0.0256  
X0.2514 Y-0.1315 Z0.1287 I0.0536 J-0.0256  
X0.1442 Y-0.0804 Z0.1222 I-0.0536 J0.0256  
X0.2515 Z0.1157 I0.0536 J-0.0256  
X0.1091 I-0.0536 J0.0256  
X0.0536 J-0.0256  
X0.0536 J-0.0256  
X0.0536 J-0.0256  
X0.0536 I-0.0536 J0.0256  
X0.0536 I-0.0536 J0.0256  
X0.0536 I-0.0536 J0.0256  
X0.0536 I-0.0536 J0.0256  
X0.2543 Y-0.0877 Z0.075 I-0.0536 J0.0256  
X0.1751 Y-0.1133 I-0.0396 J-0.0128  
G2 X0.1827 Y-0.144 I-0.2199 J-0.0709 F40.  
X0.1871 Y-0.1796 I-0.3322 J-0.0587  
G3 X0.1951 Y-0.1798 I0.004 J0.0003  
X0.1957 Y-0.134 I-0.2397 J0.026
```



Lesson 2

Shop Safety

Upon successful completion of this lesson, you will be able to:

- Display an awareness of shop safety.
- Apply general shop safety rules.
- Select appropriate shop clothing.
- Demonstrate safe personal conduct in the shop.
- Apply additional safety precautions when running a new CNC program.

Overview

This is the most important lesson of this course. Machine shops are inherently dangerous environments. What you do not know can seriously injure you. The purpose of this lesson is to provide specific safety rules but also to develop an attitude of safety awareness. This awareness should lead to wariness; which is your best defense against injury. Nurture this wariness. Develop a habit of always working in a thoughtful, methodical and deliberate way.

Never forget: **No project or deadline is worth risking serious injury.** Don't let a moment of inattention or neglect adversely impact the rest of your life.

Safety First

2.1 – Safety Awareness

Most machinists go their entire career with no serious injury even though they work with many different machines that expose them to risks. Machinists are, by nature and training, careful and methodical. They learn from experience a respect for equipment and attitude of safety. Knowing that ignorance can hurt you is essential to cultivating an attitude of safety.

It is true that CNC machines are, generally speaking, safer than manual machine tools. They are usually completely enclosed which reduces the risk of flying chips, debris from broken tools, or contact with a spinning tool. Yet machine shops are inherently dangerous places that are unforgiving of any carelessness, ignorance, or neglect.

Cutting tools, and the chips they produce, are sharp. Chips ejected from the machine can cause eye injuries. CNC machines can move over one foot in less than a second. Any physical contact with a spinning tool will result in serious cuts or loss of a limb. Remember, if it can cut metal it can certainly cut skin and bone.

Here are some examples where a failure to know or apply shop safety rules caused injury:

- A person forgets to wear safety glasses and sustains an eye injury from a metal chip thrown over the top cover of a CNC mill as they walk through the shop.
- A person leans against a bench where a cutting chip has fallen, resulting in a cut to their hand.
- A person wearing open sandal shoes has a chip fall between their foot and shoe, causing a cut.
- A person leaning over a machine suddenly raises their head and bumps into a tool stored in the tool changer, causing a severe cut.
- A person reaches into the machine to remove a part, gets distracted and rakes their arm against an end mill.
- A person grinds a piece of aluminum on a bench grinder with a stone type wheel. The aluminum embeds in the porous wheel and expands due to heating, causing the wheel to explode and throw off fragments at high speed.

2.2 – Personal Conduct & Shop Etiquette

It is important to follow strict rules of personal conduct and etiquette in the shop. This will keep you and your peers safe and promote a professional and hospitable environment:

- Know where your hands are at all times. Pockets are a great place for idle hands.
- Move deliberately and always look where your hands are going.
- Always be aware of what could happen if your hand slips. For example, when tightening a bolt, think, “What would happen if the wrench slipped? Would my hand or arm contact a tool or pile of sharp chips?”
- Always be aware of what could happen if you slipped and lost your footing and fell. Would your momentum or center of gravity cause you to fall into a sharp tool or other hazard?
- No horseplay or practical jokes are allowed in the shop. None. Ever.
- Be considerate. Do not engage in loud or unnecessarily talk.
- Do not interrupt others working at the machine. This could cause them to make a mistake.
- Never borrow tools from a private tool box without first asking the owner. If they refuse, accept it graciously.
- Respect professional Machinists. There is much you can learn from them.
- Do not make unreasonable demands (“I need it yesterday” etc.).
- Clean up after yourself. Leave the machine and surrounding area at least as clean as you found it.
- Always put tools and equipment where you found them.

2.3 – Shop Clothing

Follow these rules of personal dress for the shop:

- Wear ANSI approved safety glasses or ANSI safety approved glasses with side shields. You must wear safety glasses at all times in the shop, not just when at the machine.
- If machining operations are loud, use hearing protection.
- Do not wear flip flops or sandals. Leather shoes are best. Steel toe shoes are not necessary unless handling heavy objects that would crush regular shoes.
- Do not wear long sleeve shirts because these could get caught in equipment. Wear short sleeves or T-shirts.
- Remove rings and watches when at the machine.
- Do not wear short pants. Wear sturdy long pants like blue jeans or work pants.
- Long hair should be tied back or under a hat to prevent it being caught in the machine spindle.
- Never wear gloves as they can be caught in the machine. Latex gloves are acceptable.

Figure 1: Proper Machine Shop Attire

2.4 – General Safety Practices

Rules for a safe workplace:

- Never use any equipment which you have not been trained to operate by a qualified person.
- Never tamper with a machine safety guard or switch.
- Get into the habit of constantly tidying the workspace. A clean workplace is safer. Random metal can not only scratch and ruin finished parts, they can cause severe cuts.
- Use caution when handling cutting tools. They are very sharp. Never handle a tool by its cutting flutes.
- Never start or jog the machine until you have checked that the work area is clear.

- Never push the start button on the machine unless you are certain your setup is capable of safely holding the part against all cutting forces during machining.
 - Use caution when running a new program: especially at the start of program and after a tool change.
 - Know where the emergency stop is on the machine and practice using it before you need it.
 - Never run a machine alone or without other people within hearing distance.
 - When working with someone else at the machine, clearly communicate who is running the machine.
 - Never have one person touching the control while the other is working in the machine envelope.
 - Use a paint brush to sweep away sharp chips. Never use your hands or a rag.
 - Never use an air hose to clear chips from a machine. Flying chips are dangerous to you and others.
 - Liquids spills are slipping hazards. Clean spills immediately.
 - Dirty or oily rags must be stored in a fireproof canister. These can spontaneous combust and cause a fire.
 - Lift with your legs, not your back.
 - Never lift anything more than you can comfortably handle.
 - Get help handling heavy or bulky objects.
 - At the end of the program, command the machine to position the part close to the operator so it can be easily reached without leaning far into the CNC machine.
 - Never leave a running machine unattended.
 - Before shutting the machine down, remove any tools from the spindle.
 - Avoid contact with coolant. Water-based coolant contains microbes that can cause infection.
 - Immediately treat and cover even minor cuts.
 - Report any injuries immediately.
 - Remain alert. Think safety in everything you do.
-

2.5 CNC Safety Practices

Use these extra precautions when running a CNC program for the first time:

- Use machine Rapid and Feed override controls to slow the machine down.
- A major cause of crashes is setting the tool or fixture offset incorrectly. Pay particular attention to moves at the start of program and immediately after a tool change as the tool moves towards the part. Use single-block mode to advance through the program one line at a time until the tool is at cutting depth.
- Remain at the machine with a hand on or near the emergency stop button.
- Stop machine motion at the first sign of trouble.

2.5 Safety Contract

Training facilities should require everyone to pass a safety quiz and sign a safety contract before allowing work in the shop. The safety contract makes clear the obligations and operating regulations of the facility. Failure to abide by the terms of the contract is cause for dismissal from the shop. An example safety contract is shown on the facing page.

Warning

What you don't know CAN hurt you.

**Never operate a machine that you have not
been properly trained to use by a qualified person.**

**Read the machine operator manuals
and follow all safety instructions.**

THINK SAFETY FIRST in all you do.

Lesson 3

CNC Tools

Upon successful completion of this lesson, you will be able to:

- List the most commonly used CNC tools.
- Determine spindle rotational direction.
- Interpret a chip formation diagram.
- Define chip load.
- Distinguish between climb and conventional milling.
- Compute cutting speeds and feeds for a specified tool, material and operation using reference tables.

Overview

A wide range of tool types and configurations are available for CNC milling machines. Discussing every type, variation and use is beyond the scope of this course. This chapter introduces the most commonly used tools for prototype and short run production machining. Any tool supply catalog will list many others.

- End mills (Flat, Ball, Bull and Chamfer) Face mill
- Corner Rounding tools
- Slot Tools
- Spot-Center Drill
- Twist Drill
- Tap
- Reamer
- Counterbore

3.1 - End Mills

Milling tools include flat, ball, bull nose and chamfer.

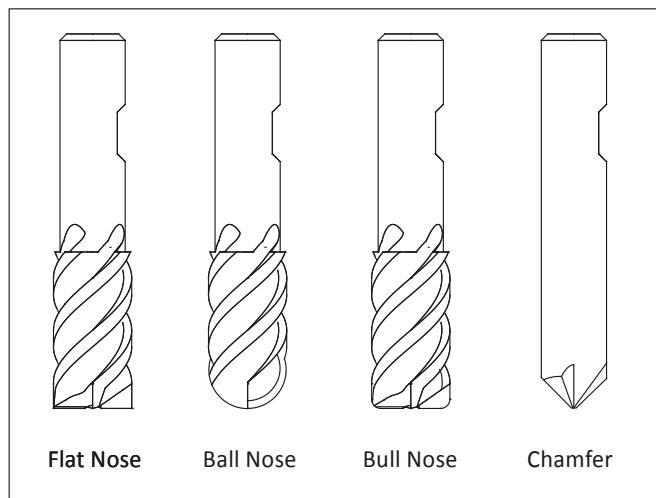


Figure 1: Mill Tool Nose Types

Flat nose mills are used for milling 2D contours and pockets. **Ball nose** mills are used for 3D milling. **Bull nose** end mills have a radius corner. They are used to create a fillet on the bottom of a wall. Because they are sturdier than an end mill they are also sometimes used for roughing operations. **Chamfer mills** have an angled nose used to create a chamfer or to de-burr parts.

Number of Flutes

Milling tools usually have either two or four cutting flutes. Two flute cutters provide more chip clearance when milling in close areas. Four flute mills are more rigid, can be fed faster, and are preferred when greater chip clearance is not required, such as when milling an outside contour.

Center-Cutting End Mills

Milling tools are either center cutting or non-center cutting. Center cutting mills can plunge straight down into material, while non-center cutting tools cannot.

Figure 2 below shows the cutting end view of a center cutting and non-center cutting end mill. Notice that the cutting edges of the center cutting end mill continues to the center of the tool. The center of the other has a small hole at the center. Non-center cutting end mills require a pilot hole, ramping or helical motion to plunge into material.

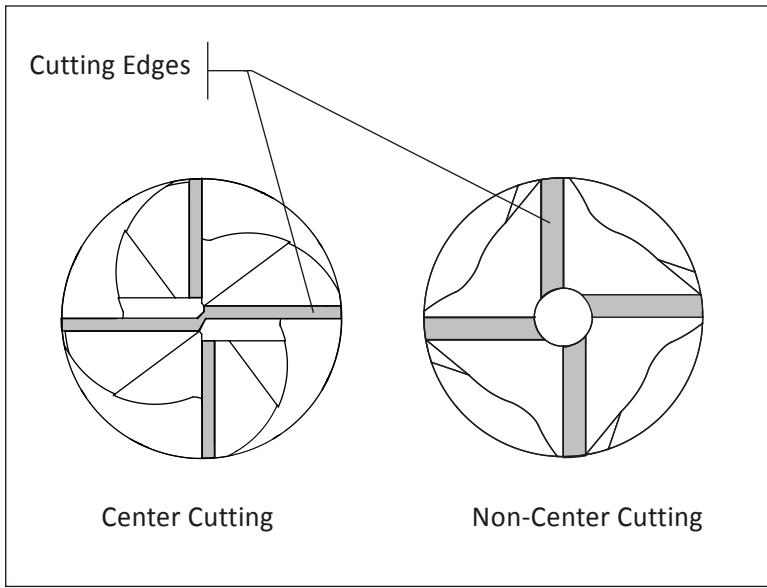


Figure 2: End View of Center and Non-Center Cutting End Mill

3.2 - Face Mill

A face mill has cutting inserts that are replaced when worn. They are rigid, may have up to eight or more cutting edges, and can remove material quickly. They are often used for the first machining operation to quickly create a flat finished face on the part.

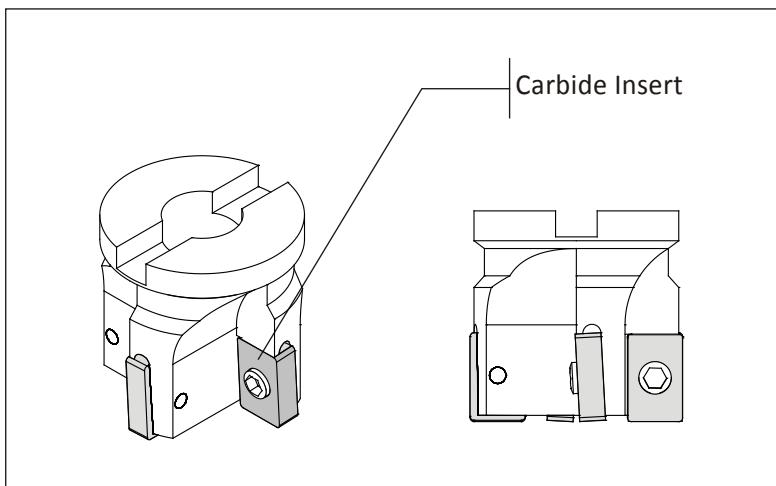


Figure 3: Face Mill

3.3 - Corner Radius Tool

Corner radius (also called Corner Round) tools are used to place a fillet on the outside corner of a part.

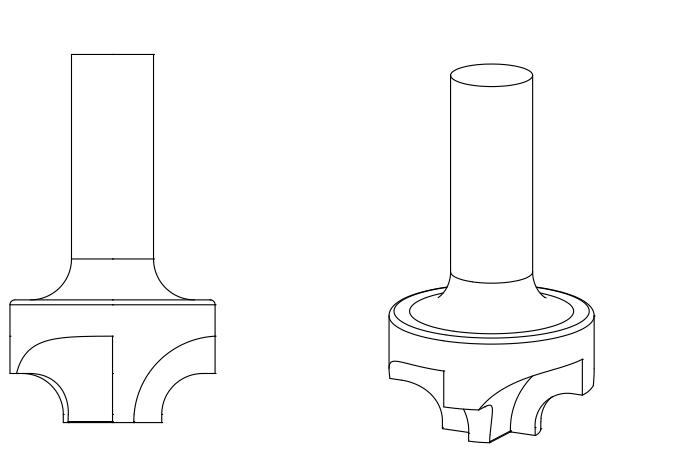


Figure 4: Corner Round Tool

3.4 - Slot Mill/Slotting Saw

Slot mills include side milling cutters, slitting saws, and Woodruff keyset cutters. Slitting saws and side milling cutters are installed on a special arbor. Woodruff cutters are single piece tools used for creating slots and undercuts that can be held in a standard tool holder.

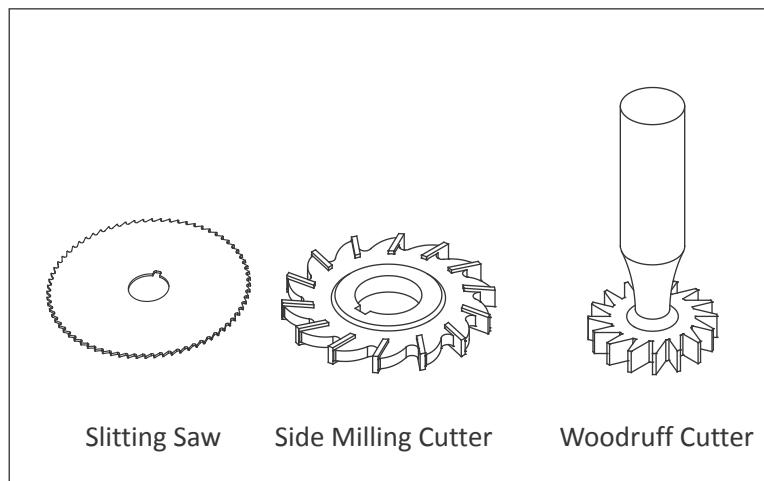


Figure 5: Slot Tools

3.5 Hole-Making Tools

Center-Spot Drills

Center (spotting) drills are short and very rigid drills used to create a conic on the face of the part. Because they come to a sharp point and resist bending, they locate the hole precisely. The conic helps prevent the subsequent drill from wobbling and ensure the drill is located precisely and drills straight down.

Countersink drills are used to create the conical face for a machine screw. Combined spotting-countersinks are used to create a screw clearance hole and countersink in one operation.

There are many different sizes and tip angles of center, countersink, and combined drills. Be sure the tip angle of the countersink matches the included angle of the machine screw, and that the drill diameter is greater than the screw head diameter.

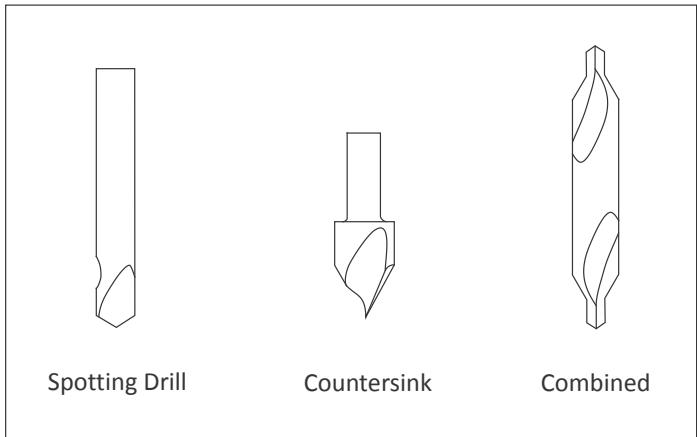


Figure 6: Countersink and Center Drill

Twist Drill

Twist drills are available in many diameters and lengths. Usually made of high speed steel, carbide, or cobalt, they may also be coated with titanium nitride (TiN) for longer life. The tip angle of most twist drills is 118 degrees.

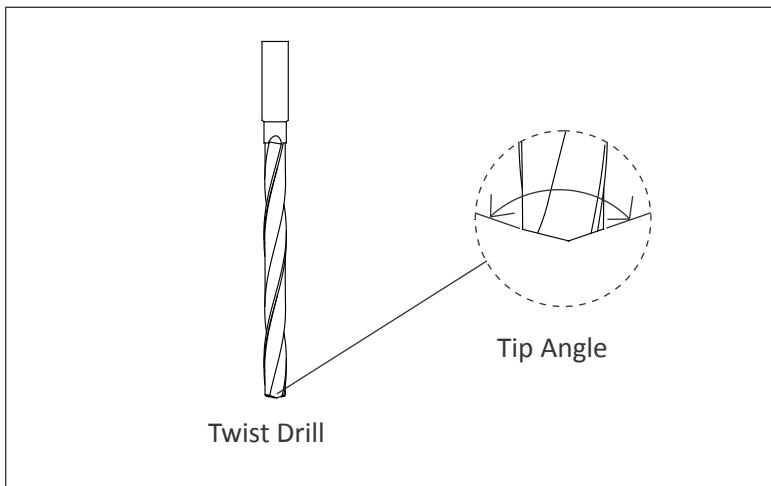


Figure 7: Twist Drill

Taps

Cutting taps form threads by shearing material away. **Form taps** (roll taps) form the thread by forming the metal to shape. Form taps produce no chips and are used for soft materials including aluminum, copper, brass and plastics.

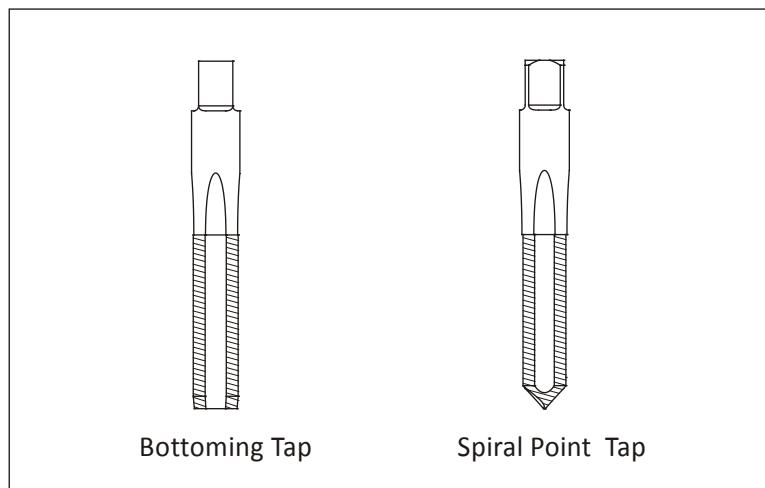


Figure 8: Taps

Bottoming taps are used to tap blind holes. **Spiral point taps** push the chip ahead and out the bottom of a through hole.

Taps require a hole drilled to the correct size to ensure the thread is formed properly. For example, a $\frac{1}{4}$ -20 cutting tap requires drilling a .201 (#7) hole. Refer to the drill chart in Appendix A to find the correct drill size for a specified thread size and fit.

Most CNC Machines support rigid tapping, which means the tap can be held in a rigid holder. The tap is advanced at a feed rate that matches the thread lead into the hole. The spindle then stops, reverses, and backs out of the hole.

Machines without rigid tapping require special tapping attachments. Always refer to the manufacturers' instructions as the speed, feed, and other machining parameters for tapping attachments may be different than those for rigid tapping.

Reamer

Use reamers to create holes of precise shape and excellent surface finish. Reamed holes are usually accurate within .0002 inches diameter. For example, a reamer is used for holes used for ground pins and bushings.

Reamers require a specific size hole be drilled before use. Cutting speeds and feeds are also important. Remove too little or too much material and the hole will not be the correct size.

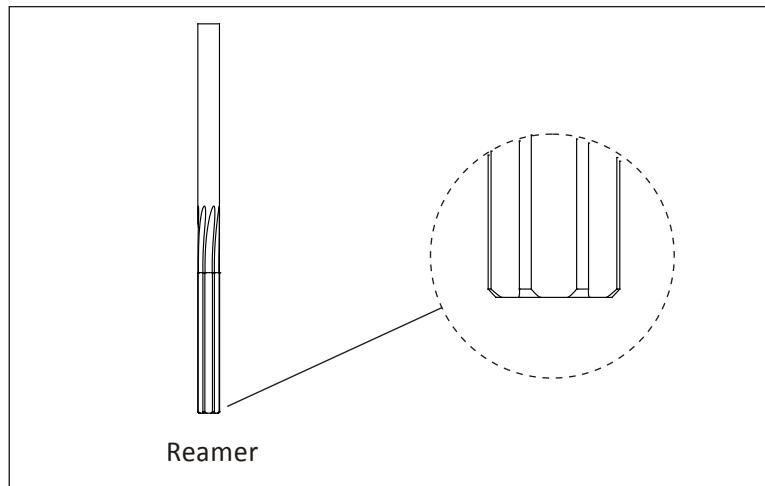


Figure 9: Reamer

Counterbore

A counterbore looks similar to a end mill with a pilot in the center. It is used to spot face holes, and the pilot ensures the spot face is centered on the hole.

Counterboring is not necessary when using a CNC machine. Rather, create a spot face using a pocket or circle mill tool path. This saves having to buy and stock counterbore tools and pilots, and the time required to load and set up the counterbore.

3.6 Cutting Tool Fundamentals

Rotation Direction

All tools (except left-handed taps) rotate clockwise (M3) when viewed from the machine spindle looking down at the part.

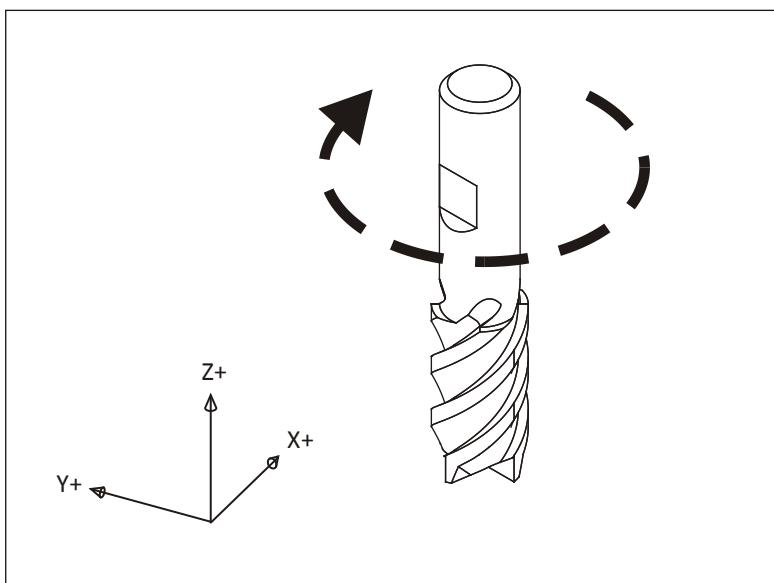


Figure 10: Clockwise Tool Rotation

Chip Formation

Cutting tools remove metal by shearing action as illustrated in Figure 11 below. As the tool advances into the material it causes a small amount of the material to shear away, forming a chip.

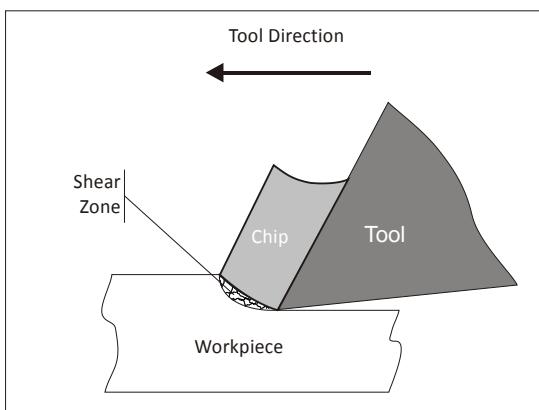


Figure 11: Chip Formation Diagram

Chip Load

The thickness of material sheared away by each cutting tooth is called the feed per tooth, or chip load. As the chip is ejected from the work area it carries with it some of the heat generated by the shearing process.

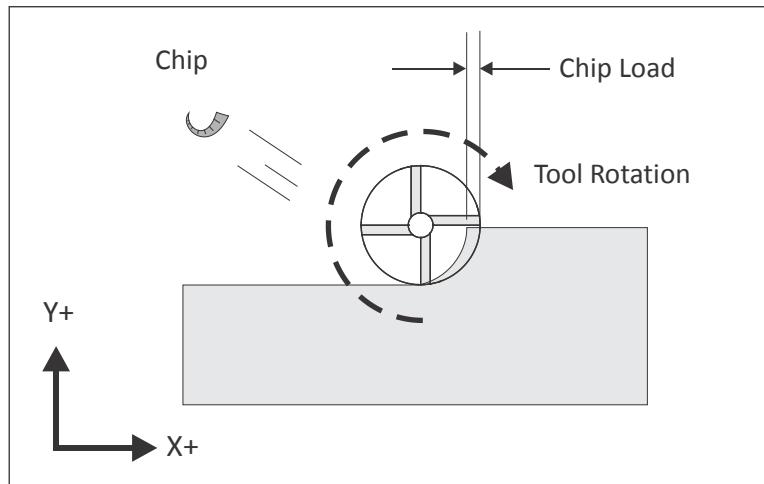


Figure 12: Chip Load

A methodology for calculating cutting speeds and feeds is presented later in this chapter. One of the best ways to validate cutting speeds and feeds is to observe the chips created by the machining process. Chips should be curled and may change color due to heating.

After gaining some experience machinists are able to adjust cutting speeds and feeds based in part on the size, shape, and color of chips and on the sound produced by the cutting process.

Climb vs. Conventional Milling

Milling tools can advance through the material so that the cutting flutes engage the material at maximum thickness and then decreases to zero. This is called **Climb Milling**.

Cutting in the opposite direction causes the tool to scoop up the material, starting at zero thickness and increasing to maximum. This is called **Conventional Milling**.

Conventional milling is used often on manual machines because backlash in the machine lead screws causes the tool to lurch when climb cutting. This is not a problem on CNC machines because they use ball screws.

Conventional milling causes the tool to rub against the cutting surface, work hardening the material, generating heat, and increasing tool wear. Raking chips across the finished surface also produces a poorer surface finish.

Unless specifically recommended by the tool manufacturer for the material being milled, always use climb milling on a CNC. Climb milling produces far less cutting pressure and heat, leaves a better surface finish, and results in longer tool life.

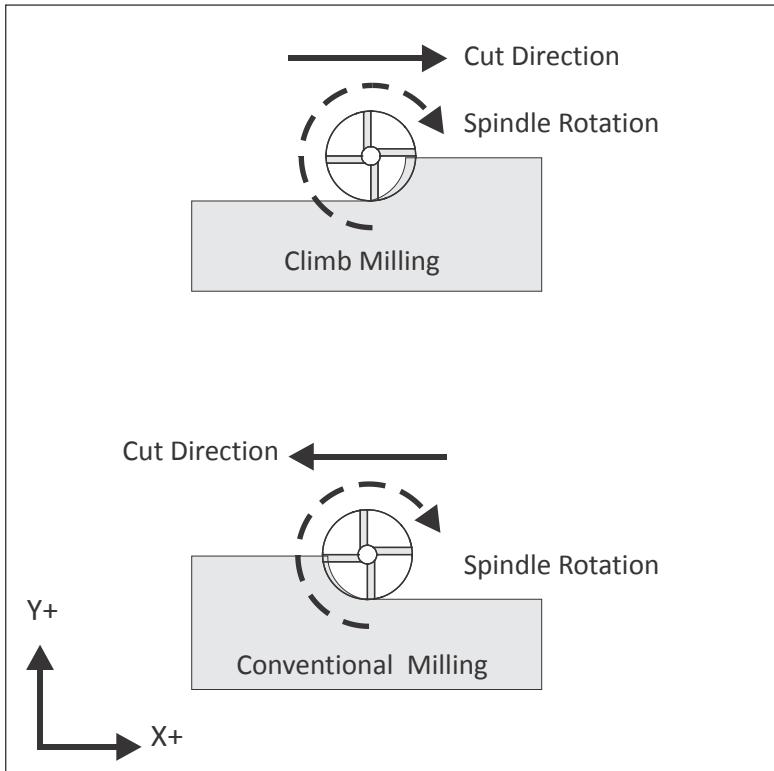


Figure 13: Climb vs. Conventional Milling

3.7 Cutting Speeds and Feeds Formulas

The tool moves through the material at a specified rotational speed, defined in revolutions per minute (RPM), and feed rate, defined in inches per minute (IPM). Probably the most vexing problem for the beginning CNC machinist is selecting proper cutting speeds and feeds. This selection is actually more difficult on a CNC than a manual mill because, with a manual mill, the operator can feel the cutting pressure and alter the feed based in part on the cutting force.

CNC mills require calculating speeds and feeds in advance. These speeds and feeds can, and often are, adjusted at the machine based on chip shape and color, cutting sound, and machine horsepower meter readings.

The best source of data about cutting speeds and feeds for a specific tool, application, and material is the tool supplier. Much of this data is found on manufacturer's web sites or printed tooling catalogs. Tool sales representatives can be a valuable resource, so if you do a lot of machining, develop a good relationship with a knowledgeable representative.

Another source of speeds and feeds data is CAD/CAM software. These have become increasingly sophisticated and often provide good cutting data.

Yet even the best speed and feed data is just a starting point. Speeds and feeds require adjustment due to many factors including the maximum spindle speed or horsepower of the machine, rigidity of work holding, and the quality and condition of the machine tool itself.

The following pages provide cutting data for the most commonly machined materials and a methodology for calculating speeds and feeds. As always, use common sense. If the part is held by double sided tape, feeds based on vise work holding are probably too high. If the tool is very long and thin, speeds and feeds will likely require reduction.

Speed Formula

Milling machine cutting speeds are derived from the following formula:

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{\text{SFM} \left(\frac{\text{ft}}{\text{min}} \right)}{\text{Circumference}(\text{ft})}$$

Figure 14: Speed Formula

Speed is the rotational frequency of the tool (Spindle Speed) in revolutions per minute (**RPM**).

SFM (Surface Feet per Minute) is the speed at which the material moves past the cutting edge (outside diameter) of the tool in feet per minute. SFM values depend on the tool type, tool material, and material being machined.

Circumference is the circumference of the cutting tool in feet.

How Speed Formula is Derived

Because cutting tools are defined by their diameter in inches, this formula is rewritten and simplified as follows:

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{\text{SFM} \left(\frac{\text{ft}}{\text{min}} \right)}{\pi \times \left(\frac{1\text{ft}}{12\text{in}} \right) \times \text{Dia}(\text{in})}$$

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{\text{SFM} \left(\frac{\text{ft}}{\text{min}} \right)}{\frac{\pi}{12} \times \text{Dia}(\text{in})}$$

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{\text{SFM} \left(\frac{\text{ft}}{\text{min}} \right) \times 3.82}{\text{Dia}(\text{in})}$$

Figure 15: Speed Formula (Simplified)

DIA is the tool diameter in inches.

3.82 is a constant derived from $12/\pi$ which converts the tool circumference in feet to diameter in inches.

Feed Formula

Cutting feeds are in (IPM) and use the following formula:

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = \text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) \times CL \left(\frac{\text{in}}{\frac{\text{rev}}{\text{flute}}} \right) \times NumFlutes$$

Figure 16: Feed Formula

Feed is the linear feed of the tool through the material in inches per minute.

Speed is the result of the speed formula (Figure 15) in revolutions per minute.

CL is the chip load, or how much material each cutting edge of the tool removes per revolution. Chip load is sometimes referred to as feed per tooth (**FPT**) or inches per rev (**IPR**).

NumFlutes is the number of cutting flutes. (For a twist drill, this value is one.)

Tap Feed Formula

For tapping operations, feed rate is based on the number of threads per inch and feed rate:

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = \frac{\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right)}{\text{TPI}}$$

Figure 17: Tap Feed Formula

Feed is the linear feed of the tool through the material in inches per minute.

Speed is the result of the previous formula in revolutions per minute.

TPI is the threads per inch of the tap. For example, the TPI of a 1/4-20 tap is 20.

3.8 - Speed/Feed Examples

Milling Speed/Feed Example

Problem: Calculate the cutting speed and feed for a milling operation given the following values:

Parameter	Value
Tool Diameter	.500in
NumFlutes	4
SFM	600ft/min
IPR	.005in

Table 1: Mill Speed/Feed Example

Solution:

Step 1: Calculate RPM

$$Speed(\frac{rev}{min}) = \frac{\left(SFM(\frac{ft}{min}) \times 3.82\right)}{Dia(in)}$$

$$Speed(\frac{rev}{min}) = \frac{600(\frac{ft}{min}) \times 3.82}{.500in}$$

$$Speed(\frac{rev}{min}) = 4584 (\frac{rev}{min})$$

Step 2: Calculate IPM

$$Feed(\frac{in}{min}) = Speed(\frac{rev}{min}) \times CL \left(\frac{in}{rev} \right) \times NumFlutes$$

$$Feed(\frac{in}{min}) = 4584 (\frac{rev}{min}) \times .005 \left(\frac{in}{rev} \right) \times 4Flutes$$

$$Feed(\frac{in}{min}) = 91 (\frac{in}{min})$$

Note: Round off milling speeds and feeds to the nearest integer.

Drill Speed/Feed Example

Problem: Calculate the cutting speed and feed for a drill operation given the following values:

Parameter	Value
Tool Diameter	.201in
SFM	250ft/min
IPR	.002in

Table 2: Drill Speed/Feed Example

Solution:

Step 1: Calculate RPM

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{\left(\text{SFM} \left(\frac{\text{ft}}{\text{min}} \right) \times 3.82 \right)}{\text{Dia}(\text{in})}$$

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{250 \left(\frac{\text{ft}}{\text{min}} \right) \times 3.82}{.201 \text{in}}$$

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = 4751 \left(\frac{\text{rev}}{\text{min}} \right)$$

Step 2: Calculate IPM

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = \text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) \times \text{CL} \left(\frac{\frac{\text{in}}{\text{rev}}}{\text{flute}} \right) \times \text{NumFlutes}$$

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = 4751 \left(\frac{\text{rev}}{\text{min}} \right) \times .002 \left(\frac{\frac{\text{in}}{\text{rev}}}{\text{flute}} \right) \times 1 \text{Flutes}$$

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = 9.5 \left(\frac{\text{in}}{\text{min}} \right)$$

Note: Round off drilling feeds to the first decimal point.

Tap Speed/Feed Example

Problem: Calculate the cutting speed and feed for a $\frac{1}{4}$ -24 tap operation given the following values:

Parameter	Value
Tool Diameter	.25in
SFM	100ft/min
TPI	24

Table 3: Tap Speed/Feed Example

Solution:

Step 1: Calculate RPM

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{\left(\text{SFM} \left(\frac{\text{ft}}{\text{min}} \right) \times 3.82 \right)}{\text{Dia}(\text{in})}$$

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{100 \left(\frac{\text{ft}}{\text{min}} \right) \times 3.82}{.25\text{in}}$$

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = 1528 \left(\frac{\text{rev}}{\text{min}} \right)$$

Step 2: Calculate IPM

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = \frac{\left(\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) \right)}{\text{TPI}}$$

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = \frac{\left(1528 \left(\frac{\text{rev}}{\text{min}} \right) \right)}{24}$$

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = 63.667 \left(\frac{\text{in}}{\text{min}} \right)$$

Note: Round off tapping feeds to three decimal points or the maximum number the machine allows.

Maximum Spindle Speed Example

In cases where the calculated spindle speed exceeds the machine capabilities, program the maximum spindle speed of the machine and use this value in the feed calculation.

Problem: Calculate the cutting speed and feed for a milling operation given the following values:

Parameter	Value
Tool Diameter	.125in
NumFlutes	2
SFM	300ft/min
IPR	.003in/rev
Maximum RPM	7,500rev/min

Table 4: Maximum Spindle Speed Example

Solution:

Step 1: Calculate RPM

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{\left(\text{SFM} \left(\frac{\text{ft}}{\text{min}} \right) \times 3.82 \right)}{\text{Dia}(\text{in})}$$

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{300 \left(\frac{\text{ft}}{\text{min}} \right) \times 3.82}{.125\text{in}}$$

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = 9168 \left(\frac{\text{rev}}{\text{min}} \right) = 7500 \left(\frac{\text{rev}}{\text{min}} \right)$$

Step 2: Calculate IPM

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = \text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) \times \text{CL} \left(\frac{\text{in}}{\text{rev}} \right) \times \text{NumFlutes}$$

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = 7500 \left(\frac{\text{rev}}{\text{min}} \right) \times .003 \left(\frac{\text{in}}{\text{rev}} \right) \times 2\text{Flutes}$$

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = 45 \left(\frac{\text{in}}{\text{min}} \right)$$

3.9 – Cutting Data

Tables on the following pages provide basic speed, feed and cutting data for some of the materials commonly used for prototypes. Use the tool manufacturer's data instead whenever it is available.

Mill Cutting Speeds (SFM) <i>surface ft/min</i>		
Material	HSS	Carbide
Aluminum	600	800
Brass	175	175
Delrin	400	800
Polycarbonate	300	500
Stainless Steel (303)	80	300
Steel (4140)	70	350

Table 5: Milling Speed Data (SFM)

Drill Cutting Speeds (SFM) <i>surface ft/min</i>				
Material	Drilling	C-Sink	Reamer	Tap
Aluminum	300	200	150	100
Brass	120	90	66	100
Delrin	150	100	75	100
Polycarbonate	240	160	120	100
Stainless Steel (303)	50	35	25	35
Steel (4140)	90	60	45	35

Table 6: Drill Cycles Speed Data (SFM)

Tip: Never use tools that have been used to machine metal to cut plastic. The sharp edge of the tool will be compromised and cutting performance and finish will suffer. A good practice is to keep two sets of tools: one for plastic and one for metal.

Tip: High speed steel cutters work best for plastics. Carbide cutters work better for aluminum and other metals.

Cutting Feeds (IPR) <i>in/rev</i>					
Operation	Tool Diameter Range (<i>in</i>)				
	<.125	.125-.25	.25-.5	.5-1.	>1.
Milling					
Aluminum	.002	.002	.005	.006	.007
Brass	.001	.002	.002	.004	.005
Delrin	.002	.002	.005	.006	.007
Polycarbonate	.001	.003	.006	.008	.009
Stainless Steel (303)	.0005	.001	.002	.003	.004
Steel (4140)	.0005	.0005	.001	.002	.003
Drilling	.002	.004	.005	.010	.015
Reaming	.005	.007	.009	.012	.015

Table 7: Feed Data (IPR)

Best Practice Machining Parameters

Best practice machining parameters for prototype and short-production milling are different than for mass production. Production machining is obsessed with minimizing run time and maximizing tool life because even small improvements per part can result in significant cost savings.

Prototype and short run production seeks to maximize reliability. Obviously, it does not make sense to risk breaking a tool or scrapping a part trying to save a few seconds if only making a few parts.

Tables 8 and 9 on the following pages list recommended machining parameters for prototypes. The values are relatively conservative and work well for materials and tool types listed on the previous pages.

For materials or tools not listed, consult cutting data from the tool manufacturer.

Recommended Machining Parameters		
Operation	Parameter	Value
All	Clearance Height	1.0 inches
All	Feed Height	.1 inches
All	Rapid Height	As needed to clear clamps and fixtures
Mill (Roughing)	Stepover (XY)	50-80% of tool dia.
Mill (Roughing)	Stepdown (Z)	25-50% of tool dia.
Drill	Peck Increment	.05 inches
Spot Drill	Dwell	.5 seconds

Table 8: Machining Parameters

Stock Finish Allowances (<i>Inches</i>)					
Operation	Tool Diameter Range (<i>in</i>)				
	<.125	.125-.25	.25-.5	.5-1.	>1.
Milling (XY)	.001	.005	.015	.020	.020
Milling (Z)	.001	.002	.005	.005	.005
Reaming	.005	.010	.012	.020	.030

Table 9: Stock Allowances

Troubleshooting Speed/Feed Problems

Do not make the mistake of thinking that the only option when encountering a machining problem is to reduce feed rate. Sometimes that is the worst thing to do and decreasing speed and increasing feed may be a better option.

Be methodical. When a problem occurs, stop. Analyze what is happening, draw on all available resources, and then devise a solution to correct the problem. The **Machinery's Handbook** (Industrial Press Inc, 2008, New York, NY, ISBN: 978-8311-2800-5) contains extensive information about diagnosing and correcting cutting tool problems. This book is an essential reference for anyone using machine tools.

Lesson 4

Coordinate System

Upon successful completion of this lesson, you will be able to:

- Identify the elements of the Cartesian coordinate system.
- Explain the purpose of the Work Coordinate System (WCS) and considerations for its selection.
- List the major elements of a closed-loop servo control mechanism.
- Identify the location and orientation of the machine coordinate system.
- Explain the purpose of the Fixture Offset XY.
- Explain the purpose of the Tool Length Offset and how to set it using a scrap a 1-2-3 block.
- Explain the purpose of the Fixture Offset Z and how it is set it using a 1-2-3 block and dial indicator.
- Name the two systems of units used in programming CNC machines

Overview

CNC motion is based on the Cartesian coordinate system. A CNC machine cannot be successfully operated without an understanding of how coordinate systems are defined in CAM and CNC machine and how the systems work together.

This lesson begins with a review of the Cartesian coordinate system and then explains in detail how the coordinate systems between CAM and the CNC machine are related to each other. It also describes how the machine work coordinate system (WCS) is set on the CNC machine so the machine knows where the part is located within the work space.

It concludes with a discussion of tool length and diameter offsets. Length offsets are required to account for different tool lengths (how far a particular tool extends out of the holder). Diameter offsets are the key to highly precise machining where part tolerances can be maintained to an accuracy of .005 inches or less.

4.1 – Cartesian Coordinate System

CNC motion is based on a 3D Cartesian coordinate system.

Number Line

The basis of this system is the number line marked at equal intervals. The axis is labeled (X, Y or Z). One point on the line is designated as the Origin. Numbers on one side of the line are marked as positive and those to the other side marked negative.

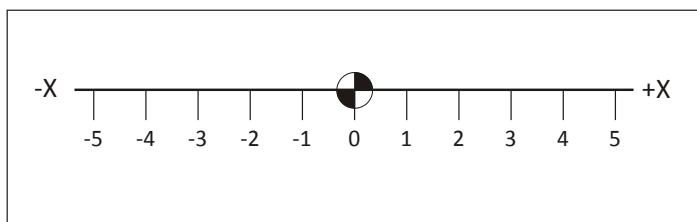


Figure 1: X-Axis Number Line.

3D Cartesian Coordinate System

The Cartesian coordinate system consists of three number lines, labeled X, Y and Z, set at 90 degree angles to each other as shown in Figure 2 below. The origin, or Datum, is where the three axes cross each other. The labels, orientations, and directions of the Cartesian coordinate system in Figure 2 are typical of most Vertical Machining Center (VMC).

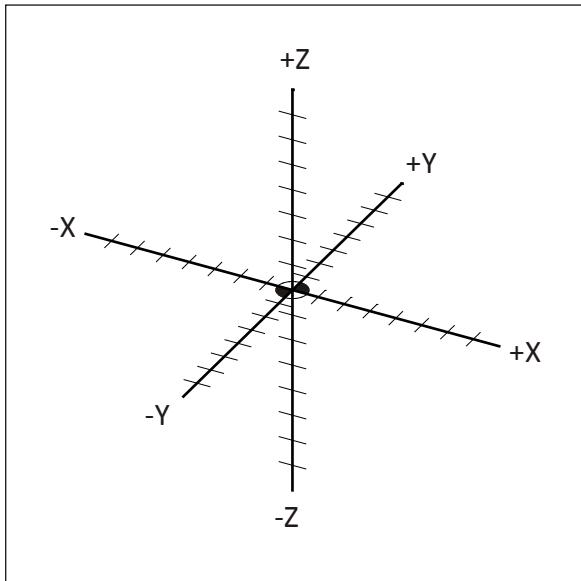


Figure 2: 3D Cartesian Coordinate System

Quadrants

Any two axes form a plane. Planes are named by the axes that define them. For example, Figure 3 shows the XY plane, which is the primary work plane for machining on a VMC. A plane can be divided into four quadrants, labeled I, II, III and IV with axes designations as shown in the illustration below.

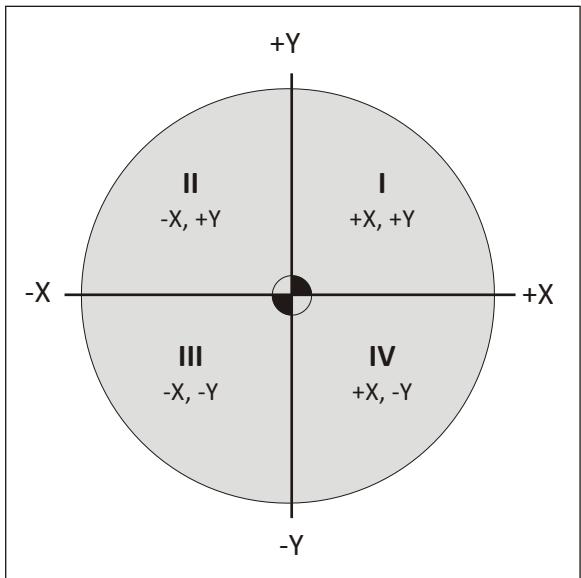


Figure 3: Quadrants

Units

CNC Programs can be written in either Inch or Metric units. The machine can be switched with a single code to accept either.

In the United States, most programming is using inch units because most tooling is in inches and machinists are more familiar with the inch measurement system. Even if the part is designed in metric, it is usually converted to inch units for machining and metric tools are used only when no inch equivalent is available (for example when creating metric tapped holes).

Table 1 lists the units and maximum precision for inch and metric data used by CNC machines.

Units and Precision			
Data Type	Inch Units	Metric Units	
Coordinate	inches	.0001	.001
Speed	rev/min	1.	rev/min
Feed	in/min	1.	mm/min
Tap Feed	in/min	.001	mm/min
			.01

Table 1: Units and Precision

4.2 Vertical Milling Center (VMC) Machine Motion

CNC machines use a 3D Cartesian coordinate system. Figure 4 shows a typical VMC with the sheet metal covers removed to expose the movable parts.

Material to be machined is fastened to the machine table. This table moves in the XY-Plane. As the operator faces the machine, the X-Axis moves the table left-right. The Y-Axis moves the table forward-backward.

The machine column grips and spins the tool. The column controls the Z-axis and moves up-down.

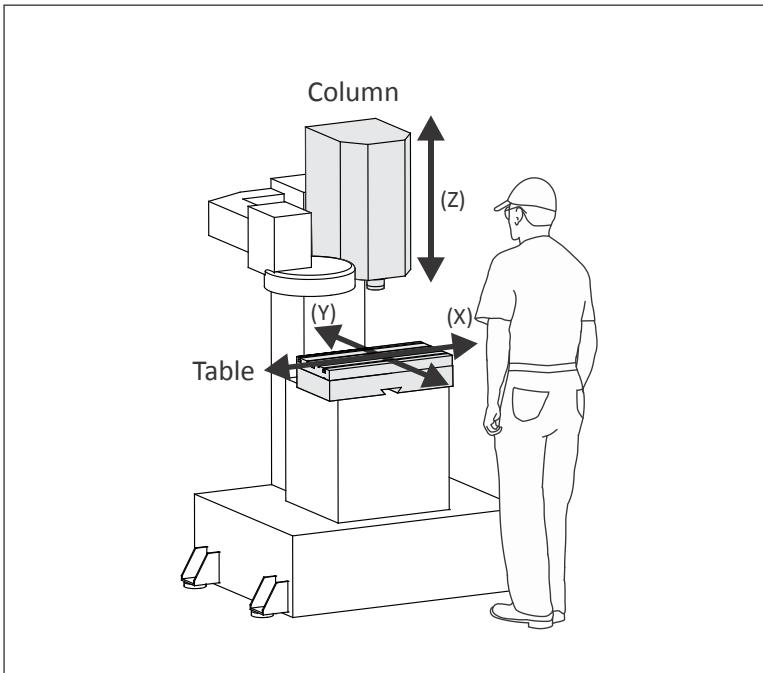


Figure 4: VMC Machine Motion

CNC Motion Control

Most CNC machines can position each axis within .0002 inches or less over the entire machining envelope. This accuracy is achieved in part by the use of a closed-loop servo mechanism, illustrated in Figure 5.

The machine control sends a motion signal, via a controller board, to a servomotor attached to each machine axis. This causes the servomotor to rotate a ball screw attached to the table or column, causing it to move. The actual position of the axis is continuously monitored and compared to the commanded position with feedback from a servo transmitter attached to the ball screw.

Ball screws have almost no backlash, so when the servo reverses direction there is almost no lag between a commanded reversing motion and corresponding change in table direction. CNC controls employ electronic compensation to adjust for any minor backlash that may exist.

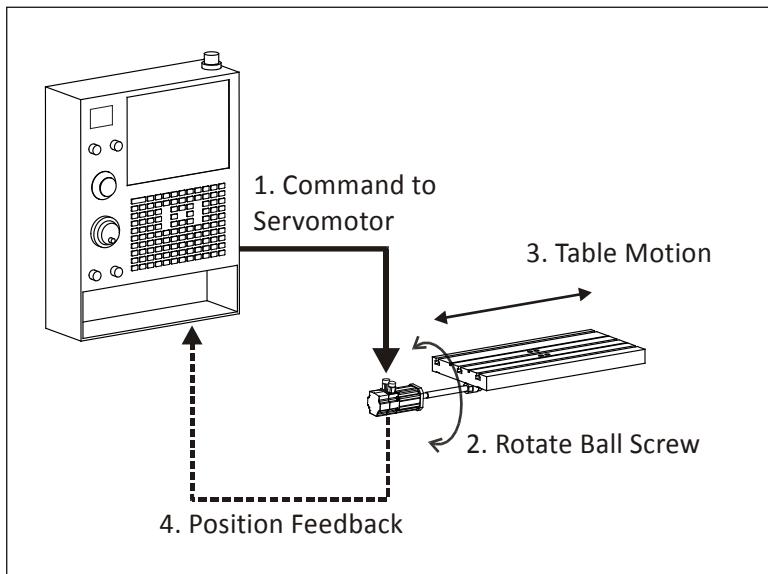


Figure 5: Closed Loop Servo Mechanism

CNC Machine Coordinates

The **CNC Machine Coordinate System** is illustrated in Figure 6. The control point for the Machine Coordinate System is defined as the center-face of the machine spindle.

The Origin point for the machine coordinate system is called **Machine Home**. This is the position of the center-face of the machine spindle when the Z-axis is fully retracted and the table is moved to its limits near the back-left corner.

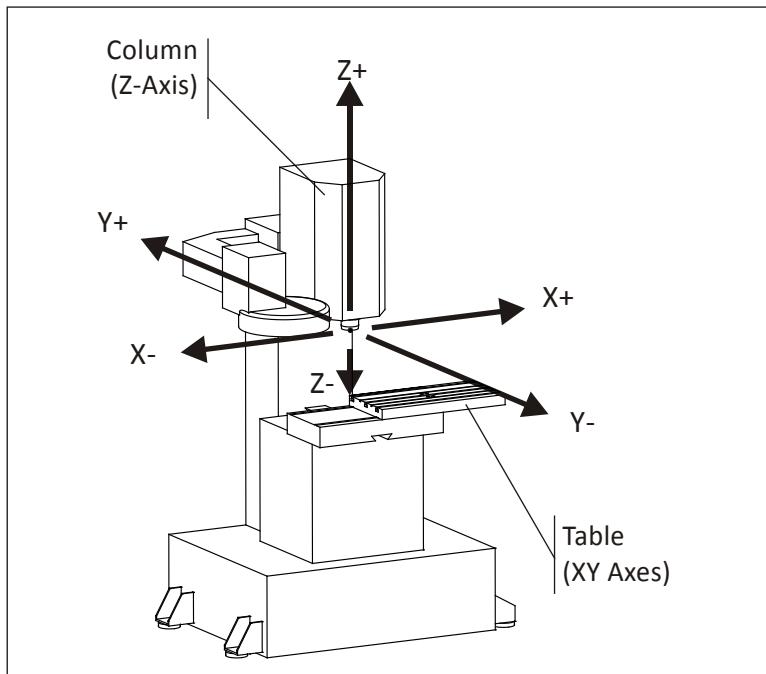


Figure 6: VMC Machine Coordinate System (At Home Position)

IMPORTANT

As shown in Figure 6, when working with a CNC, always think, work, and write CNC programs in terms of tool motion, not table motion.

For example, increasing +X coordinate values move the tool right in relation to the table (though the table actually moves left).

Likewise, increasing +Y coordinate values move the tool towards the back of the machine (the table moves towards the operator).

Increasing +Z commands move the tool up (away from the table).

About Machine Home Position

When a CNC machine is first turned on, it does not know where the axes are positioned in the work space. Home position is found by the **Power On Restart** sequence initiated by the operator by pushing a button on the machine control after turning on the control power.

The Power On Restart sequence simply drives all three axes slowly towards their extreme limits (-X, +Y, +Z). As each axis reaches its mechanical limit, a microswitch is activated. This signals to the control that the home position for that axis is reached. Once all three axes have stopped moving, the machine is said to be "homed". Machine coordinates are thereafter in relation to this home position.

4.3 Work Coordinate System

Obviously it would be difficult to write a CNC program in relation to Machine Coordinates. The home position is far away from the table, so values in the CNC program would be large and have no easily recognized relation to the part model. To make programming and setting up the CNC easier, a Work Coordinate System (WCS) is established for each CNC program.

The WCS is a point selected by the CNC programmer on the part, stock or fixture. While the WCS can be the same as the part origin in CAD, it does not have to be. While it can be located anywhere in the machine envelope, its selection requires careful consideration.

- The WCS location must be able to be found by mechanical means such as an edge finder, coaxial indicator or part probe.
- It must be located with high precision: typically plus or minus .001 inches or less.
- It must be repeatable: parts must be placed in exactly the same position every time.
- It should take into account how the part will be rotated and moved as different sides of the part are machined.

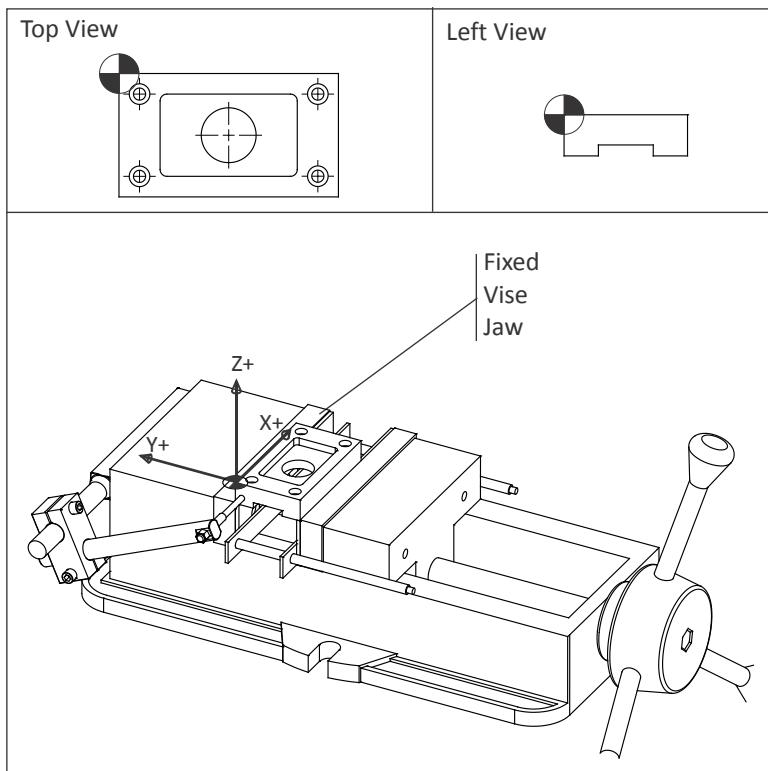


Figure 7: Work Coordinate System (WCS)

WCS Example

The following example shows why and how the WCS is set up a typical part that is machined on multiple sides by gripping in a vise. This is one of the most common ways to hold a part. Pay particular attention how the part is rotated between jobs.

Job

The term, **Job**, means a unique machining setup on the machine. For example, a part that requires the part to be moved or rotated three times on the CNC is said to be composed of three jobs; one for each setup.

WCS Example – Job 1

Figure 12 shows a part gripped in a six inch wide precision CNC vise. The outside shape of this part was machined to size on a manual mill before being set on the CNC machine. The CNC is used to create the holes, pockets, and slot on this block.

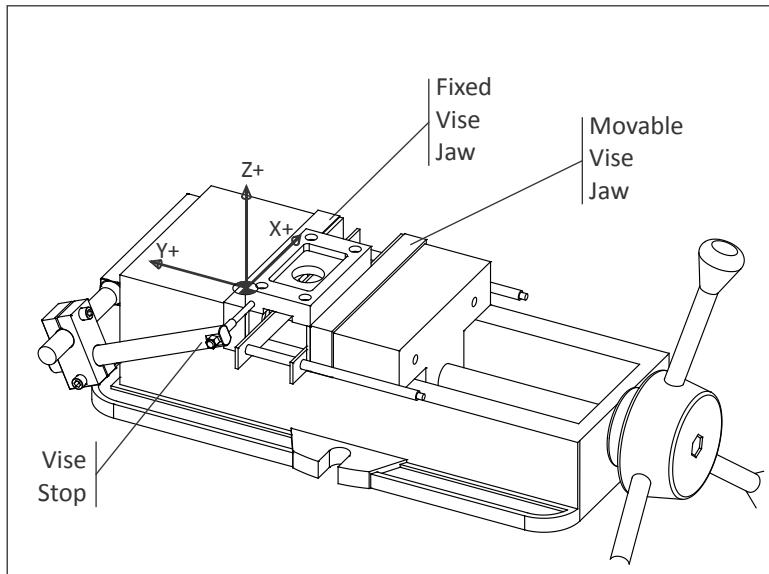


Figure 12: Tool Length Offset Example (Job 1)

Before clamping the part, the vise is aligned and bolted to the machine table. This assures the part WCS X-axis is aligned with the machine X-axis.

Parallels (precision ground rails) are used to support the part. These ensure the XY-Plane of the part is parallel to the machine table XY-Plane.

The left face of the part rests against a Vise Stop. The Vise Stop establishes the X-axis origin. As new parts are loaded into the vise, the operator slides them against the vise stop. This ensures all parts are loaded into the exact same position each time.

Because the edges of the block are already milled, the WCS XY location is easily found using an Edge Finder (Lesson 6, Set Fixture Offset XY) or part probe.

The vise has two jaws; a fixed back jaw a front jaw that can close and open to grip or release the part. Because the location of the moving jaw varies depending how much force the operator uses, it is best to locate the WCS in reference to the fixed jaw. The fixed jaw position is not significantly affected by clamping force.

Notice that, because the fixed vise jaw does not move regardless of how tightly the vise is closed, the WCS Y-origin does not change. In other words, the Y-origin is **repeatable**. The concept of repeatability is essential to precision machining. If the datum shifts for any reason, it is impossible to make any two parts exactly alike.

When using a vise, locate the WCS so the part lies in the forth quadrant: resting the Y-datum (XZ Plane) against the fixed vise jaw.

WCS Example – Job 2

After the pocket, holes, and through round hole are machined on the first side of the part, the part is flipped over in the vise to create the slot. Whether a new WCS must be defined, and how the part is flipped, depends on the part geometry and type of setup.

As shown in Figure 13, because the outside dimensions of the part were established prior to machining, and because the part is flipped 180 degrees, the location of the WCS does not change. Therefore, there is no need to define a new WCS to machine the slot. Fixture Offset G54 can be used to machine both sides.

However, how the part is turned over does matter because of how a vise works. As mentioned earlier, a vise exerts a tremendous amount of clamping force (up to 6,000 lbs or more) and so the actual position of the moving vise jaw depends on how tightly the vise is closed.

This variability is so large that it is common practice to mark the closed position of the handle with a black marker or use a torque wrench to ensure the clamping pressure is consistent between parts. Vise force can even significantly deform thin parts if excessive force is applied.

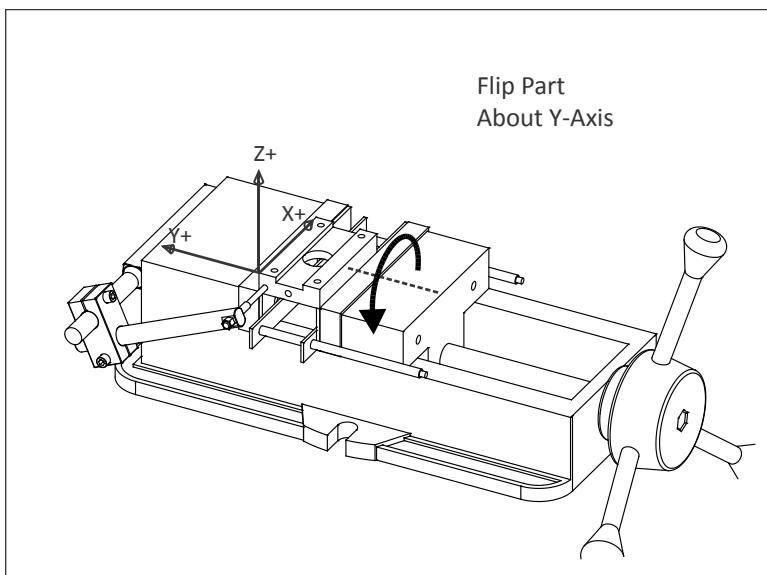


Figure 13: WCS (Job 2)

By flipping the part about the Y-axis, the same edge of the part (XZ Plane) rests against the fixed jaw. Since this position does change based on clamping force, and because the vise stop is also unaffected by clamping force, the WCS for Job 2 is also repeatable.

WCS Example – Job 3

Drilling the hole in the side of the part means turning it again to stand on end, as shown in Figure 14. Again, rotate the part about the Y-axis so that the Y-origin of the WCS (XZ reference) plane does not shift or change based on clamping force.

Notice that the WCS used in Job 1 and 2 cannot be used because the part standing on end is much taller. A new Fixture Offset is defined (G55) to shift the datum to the point shown.

It is also worth noticing that, in order to increase gripping surface, the parallels have been removed. The vise stop has also been lowered so the stylus contacts the face of the part, not the bottom of the pocket.

The best practice is to maintain as many reference surfaces as possible whenever the part is rotated. By turning the part as shown in Figure 14, two of the reference planes are used. This helps ensure the hole will be located precisely on the part side.

As a practical matter, the machinist could set up a second vise on the machine for this operation. If making many parts without a second vise, they might choose to machine the top and bottom of all parts, then reconfigure the vise as shown and make the hole in all parts.

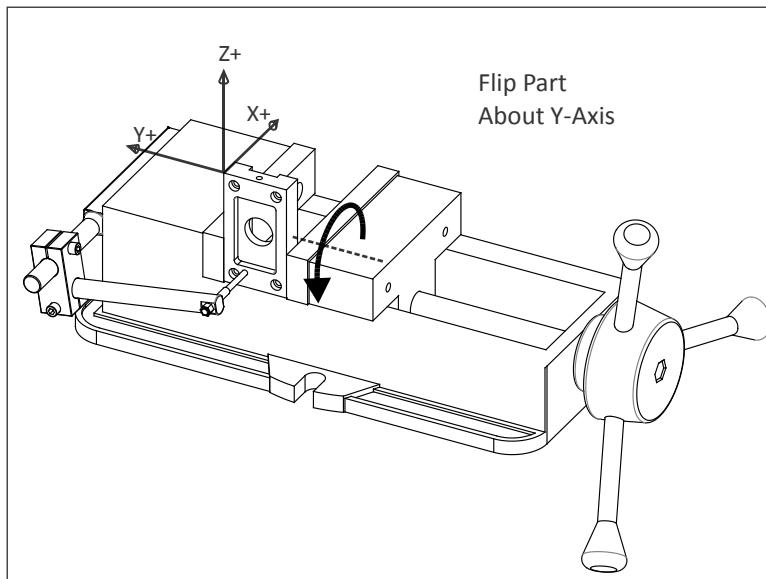


Figure 14: WCS (Job 3)

CNC machining typically involves tolerances of less than .005 inches, or about twice the thickness of a human hair. Small chips or even excess coolant under a part or vise can cause problems. Maintaining close machining tolerances requires being fastidious and consistency of work.

4.4 Machine and Tool Offsets

Machine Offsets

Because it is difficult to place a vise in the exact same position on the machine each time, the distance from Home to the WCS is usually not known until the vise is set and aligned with the machine. Machine set up is best done after the program is completely written, because it is expensive to keep a CNC machine idle waiting for the CNC programming to be done. Besides, the programmer may change their mind during the CAM process, rendering any pre-planned setup obsolete.

To complicate matters further, different tools extend out from the machine spindle different lengths, also a value difficult to determine in advance. For example, a long end mill extends further from the spindle face than a stub length drill. If the tool wears or breaks and must be replaced, it is almost impossible to set it the exact length out of the tool holder each time.

Therefore, there must be some way to relate the Machine Coordinate system to the part WCS and take into account varying tool lengths. This is done using machine **Tool and Fixture Offsets**. There are many offsets available on CNC machines. Understanding how they work and to correctly use them together is essential for successful CNC machining.

Fixture Offset XY

Fixture offsets provide a way for the CNC control to know the distance from the machine home position and the part WCS. In conjunction with Tool Offsets, Fixture Offsets allow programs to be written in relation to the WCS instead of the Machine Coordinates. They make setups easier because the exact location of the part in the machine envelop does not need to be known before the CNC program is written.

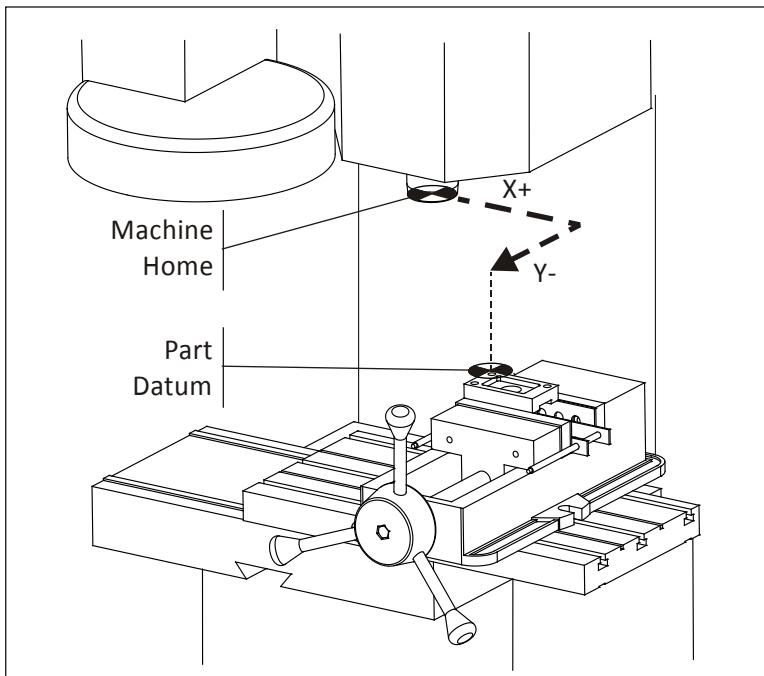


Figure 8: Fixture Offset Shifts Machine to WCS

As long as the part is positioned where the tool can reach all machining operations it can be located anywhere in the machine envelope. Once the Fixture Offset values are found, entered into the control, and activated by the CNC program, the CNC control works behind the scene to translate program coordinates to WCS coordinates.

Notice in Figure 8 how Fixture Offsets (+X, -Y) are used to shift the centerline of the machine spindle directly over the WCS.

Fixture Offset Z

The Fixture Offset Z value is combined with the Tool Length offset to indicate to the machine how to shift the Z-datum from part home to the part Z-zero, taking into account the length of the tool. Fixture Offset Z may or may not be used, depending on how the machine is set up and operated.

Lesson 6 (CNC Operation) and Appendix B (Alternate Tool Setting Methods) describe the various ways that these two values can be used. Follow the procedure in use at your facility or refer to your machine tool documentation to determine which method to use.

Tool Length Offset (TLO)

Every tool loaded into the machine is a different length. In fact, if a tool is replaced due to wear or breaking, the length of its replacement will likely change because it is almost impossible to set a new tool in the holder in exactly the same place as the old one. The CNC machine needs some way of knowing how far each tool extends from the spindle to the tip. This is accomplished using a **Tool Length Offset (TLO)**.

In its simplest use, the TLO is found by jogging the spindle with tool from the machine home Z-position to the part Z-zero position, as shown on the far left in Figure 9 below. The tool is jogged to the part datum Z and the distance travelled is measured. This value is entered in the TLO register for that tool. Problems with this method include the need to face mill the part to the correct depth before setting tools. Also, if the Z-datum is cut away (typical of 3D surfaced parts) it is impossible to set the datum should a tool break or wear and need to be replaced. All tools must be reset whenever a new job is set up. When this method is used, the Fixture Offset Z is not used, but set to zero.

The method shown in the center is much better and used in this book. All tools are set to a known Z-position, such the top of a precision 1-2-3 block resting on the machine table. This makes it very easy to reset tools if worn or broken.

A tool probe is very similar to the 1-2-3 block method, except the machine uses a special cycle to automatically find the TLO. It does this slowly lowering the tool until the tip touches the probe and then updates the TLO register. This method is fast, safe and accurate but requires the machine be equipped with a tool probe. Also, tool probes are expensive so care must be taken to never crash the tool into the probe.

Both the 2nd and 3rd methods also require the distance from the tool setting position (the top of the 1-2-3 block or tool probe) to the part datum to be found and entered in the Fixture Offset Z. The machine adds the two values together to determine the total tool length offset. A method for doing this is included in Lesson 6.

3 Ways to Set Tool Length Offset

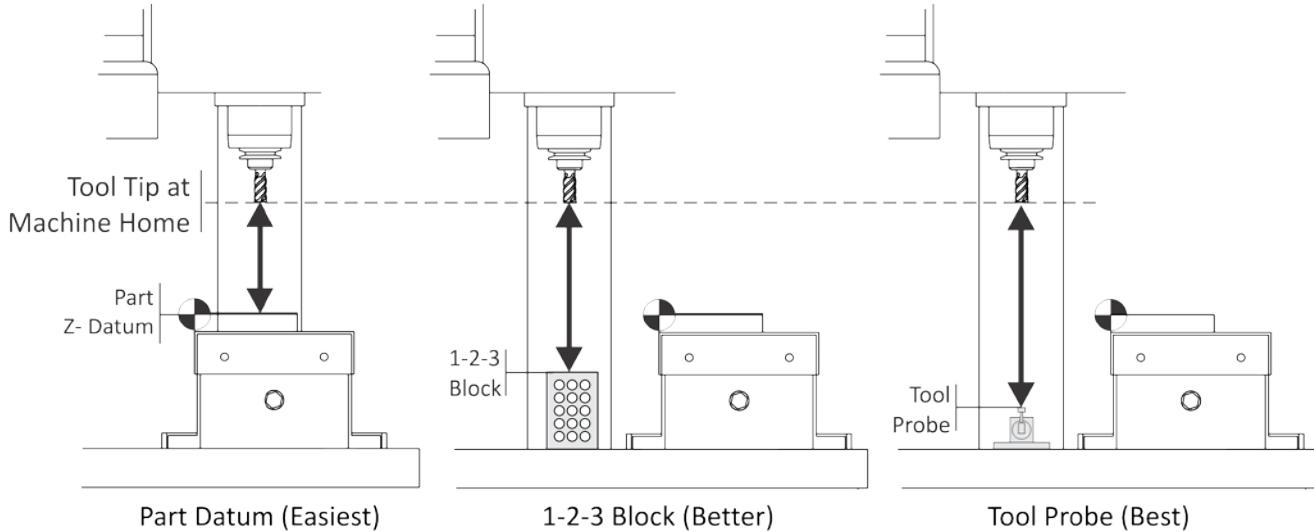


Figure 9: Ways to Set TLO

Lesson 5

CNC Programming Language

Upon successful completion of this lesson, you will be able to:

- Define CNC Program.
- List the sequence of operations in a typical CNC program.
- List the most commonly used G-codes, their meaning, and syntax.
- List the most commonly used M-codes, their meaning, and syntax.
- List the most commonly used special characters in a CNC Program and their purpose.
- Describe the organization, motion, and actions in a simple CNC program.

Overview

CNC machines are very accurate and powerful industrial robots developed jointly by Mr. John Parsons, IBM and Massachusetts Institute of Technology Servomechanism Laboratory in the 1950's.

Most CNC machine tools use a language set by the Electronics Industry Association (EIA) in the 1960's. The official name of this language is RS-274D, but everyone refers it "G-code" or "G&M Code" because many of the words of this language begin with the letters G or M.

While many of the words used by different CNC machines are the same, there are differences between makes and models. This is due in part to machines having different configurations and options. For example, a machine with a chip conveyor will have words to turn the conveyor on and off, while a machine without a conveyor does not. So, while RS-274D is a standard, it is not rigid or enforced. Always refer to the machine documentation for the exact words and syntax for your CNC machine.

Most machines have a vocabulary of at least a hundred words, but only about thirty that are used often. These thirty or so words are best memorized because they appear in almost every CNC program and knowing them helps you work more efficiently.

The G-code language was developed when machine controls had very little memory. It was therefore designed to be as compact as possible. While at first this language may seem arcane, the modern machine tool language is the safest and most efficient way yet devised to control machine tool motion. G&M codes, along with coordinates and other parameters, comprise what is called a **CNC program**.

5.1 CNC Language and Structure

CNC programs list instructions to be performed in the order they are written. They read like a book, left to right and top-down. Each sentence in a CNC program is written on a separate line, called a **Block**. Blocks are arranged in a specific sequence that promotes safety, predictability and readability, so it is important to adhere to a standard program structure.

Typically, blocks are arranged in the following order:

- 1) Program Start
- 2) Load Tool
- 3) Spindle On
- 4) Coolant On
- 5) Rapid to position above part
- 6) Machining operation
- 7) Coolant Off
- 8) Spindle Off
- 9) Move to safe position
- 10) End program

The steps listed above represent the simplest type of CNC program, where only one tool is used and one operation performed. Programs that use multiple tools repeat steps two through nine for each.

Table 1 and Table 2 on the next pages show the most common G and M codes that should be memorized if possible.

Like any language, the G-code language has rules. For example, some codes are modal, meaning they do not have to be repeated if they do not change between blocks. Some codes have different meanings depending on how and where there are used.

While these rules are covered in this chapter, do not concern yourself with learning every nuance of the language. It is the job of the job of the CAD/CAM software **Post Processor** to properly format and write the CNC program.

Program Format

The program in Figure 1 below machines a square contour and drills a hole.

Block	Description	Purpose
%	Start of program.	
O0001 (PROJECT1)	Program number (Program Name).	
(T1 0.25 END MILL)	Tool description for operator.	
N1 G17 G20 G40 G49 G80 G90	Safety block to ensure machine is in safe mode.	
N2 T1 M6	Load Tool #1.	
N3 S9200 M3	Spindle Speed 9200 RPM, On CW.	
N4 G54	Use Fixture Offset #1.	
N5 M8	Coolant On.	
N6 G00 X-0.025 Y-0.275	Rapid above part.	
N7 G43 Z1. H1	Rapid to safe plane, use Tool Length Offset #1.	
N8 Z0.1	Rapid to feed plane.	
N9 G01 Z-0.1 F18.	Line move to cutting depth at 18 IPM.	
N10 G41 Y0.1 D1 F36.	CDC Left, Lead in line, Dia. Offset #1, 36 IPM.	
N11 Y2.025	Line move.	
N12 X2.025	Line move.	
N13 Y-0.025	Line move.	
N14 X-0.025	Line move.	
N15 G40 X-0.4	Turn CDC off with lead-out move.	
N16 G00 Z1.	Rapid to safe plane.	
N17 M5	Spindle Off.	
N18 M9	Coolant Off.	
(T2 0.25 DRILL)	Tool description for operator.	
N19 T2 M6	Load Tool #2.	
N20 S3820 M3	Spindle Speed 3820 RPM, On CW.	
N21 M8	Coolant On.	
N22 X1. Y1.	Rapid above hole.	
N23 G43 Z1. H2	Rapid to safe plane, use Tool Length Offset 2.	
N24 Z0.25	Rapid to feed plane.	
N25 G98 G81 Z-0.325 R0.1 F12.	Drill hole (canned) cycle, Depth Z-.325, F12.	
N26 G80	Cancel drill cycle.	
N27 Z1.	Rapid to safe plane.	
N28 M5	Spindle Off.	
N29 M9	Coolant Off.	
N30 G91 G28 Z0	Return to machine Home position in Z.	
N31 G91 G28 X0 Y0	Return to machine Home position in XY.	
N32 G90	Reset to absolute positioning mode (for safety).	
N33 M30	Reset program to beginning.	
%	End Program.	

Figure 1: Simple CNC Program

5.2 - CNC Editor

CNC programs are simple ASCII character text files that can be viewed or edited in any text editor. Refer to the HSMWorks Editor Help files for specific instructions on how to use the Editor.

5.3 – Alphabetic & Special Character Address Codes

Every letter of the alphabet is used as a machine address code. In fact, some are used more than once, and their meaning changes based on which G-code appears in the same block.

Codes are either modal, which means they remain in effect until cancelled or changed, or non-modal, which means they are effective only in the current block.

The table below lists the most common address codes. A complete list is included in Appendix B, G-M Code Reference.

Code	Meaning
A	Rotation about X-axis.
B	Rotation about Y-axis.
C	Rotation about Z-axis.
D	Cutter diameter compensation (CDC) offset address.
F	Feed rate.
G	G-Code (preparatory code).
H	Tool length offset (TLO).
I	Arc center X-vector, also used in drill cycles.
J	Arc center Y-vector, also used in drill cycles.
K	Arc center Z-vector, also used in drill cycles.
M	M-Code (miscellaneous code).
N	Block Number.
O	Program Number.
P	Dwell time.
Q	Used in drill cycles.
R	Arc radius, also used in drill cycles.
S	Spindle speed in RPM.
T	Tool number.
X	X-coordinate.
Y	Y-coordinate.
Z	Z-coordinate.

Table 3: Common Alphanumeric Address Codes

Alphabetic Address Code Definitions

Here are the most common alphabetic address code definitions, examples and restrictions of use. Most modern machines use these codes.

A,B,C 4th/5th Axis Rotary Motion

Rotation about the X, Y or Z-axis respectively. The angle is in degrees and up to three decimal places precision.

G1 A30.513 B90.

D Tool Diameter Register

Used to compensate for tool diameter wear and deflection. D is accompanied by an integer that is the same as the tool number (T1 uses D1, etc). No decimal point is used. It is always used in conjunction with G41 or G42 and a XY move (never an arc). When called, the control reads the register and offsets the tool path left (G41) or right (G42) by the value in the register.

G1 G41 X1. D1

F Feed Rate

Sets the feed rate when machining lines, arcs or drill cycles. Feed rate can be in Inches per Minute (G94 mode) or Inverse Time (G93 mode). Feed rates can be up to three decimal places accuracy (for tap cycles) and require a decimal point.

G1 X1. Y0. F18.

G Preparatory Code

Always accompanied by an integer that determines its meaning. Most G-codes are modal. Expanded definitions of G-codes appear in the next section of this chapter.

G2 X1. Y1. I.25 J0.

H Tool Length Compensation Register

This code calls a tool length offset (TLO) register on the control. The control combines the TLO and Fixture Offset Z values to know where the tool is in relation to the part datum. It is always accompanied by an integer (H1, H2, etc), G43, and Z coordinate.

G43 H1 Z1.

I Arc Center or Drill Cycle Data

For arc moves (G2/G3), this is the incremental X-distance from the arc start point to the arc center. Certain drill cycles also use I as an optional parameter.

G2 X.1 Y2.025 I0. J0.125

J Arc Center or Drill Cycle Data

For arc moves (G2/G3), this is the incremental Y-distance from the arc start point to the arc center. Certain drill cycles also use J as an optional parameter.

G2 X.1 Y2.025 I0. J0.125

K Arc Center or Drill Cycle Data

For an arc move (G2/G3) this is the incremental Z-distance from the arc start point to the arc center. In the G17 plane, this is the incremental Z-distance for helical moves. Certain drill cycles also use J as an optional parameter.

G18 G3 X.1 Z2.025 I0. K0.125

M Preparatory Code

Always accompanied by an integer that determines its meaning. Only one M-code is allowed in each block of code. Expanded definitions of M-codes appear later in this chapter.

M8

N Block Number

Block numbers can make the CNC program easier to read. They are seldom required for CAD/CAM generated programs with no subprograms. Because they take up control memory most 3D programs do not use block numbers. Block numbers are integers up to five characters long with no decimal point. They cannot appear before the tape start/end character (%) and usually do not appear before a comment only block.

N100 T2 M6

O Program Number

Programs are stored on the control by their program number. This is an integer that is preceded by the letter O and has no decimal places.

O0002 (PROJECT 1)

P Delay

Dwell (delay) in seconds. Accompanied by G4 unless used within certain drill cycles.

G4 P.1

Q Drill Cycle Optional Data

The incremental feed distance per pass in a peck drill cycle.

G83 X1. Y1. Z-.5 F12. R.1 Q.1 P5.

R Arc Radius or Drill Cycle Optional Data

Arcs can be defined using the arc radius R or I,J,K vectors. IJK's are more reliable than R's so it is recommended to use them instead. R is also used by drill cycles as the return plane Z value.

G83 Z-.5 F12. R.1 Q.1 P5.

S Spindle Speed

Spindle speed in revolutions per minute (RPM). It is an integer value with no decimal, and always used in conjunction with M3 (Spindle on CW) or M4 (Spindle on CCW).

S3820 M3

T Tool number

Selects tool. It is an integer value always accompanied by M6 (tool change code).

T1 M6

X X-Coordinate

Coordinate data for the X-axis. Up to four places after the decimal are allowed and trailing zeros are not used. Coordinates are modal, so there is no need to repeat them in subsequent blocks if they do not change.

G1 X1.1252

Y Y-Coordinate

Coordinate data for the Y-axis.

G1 Y1.

Z Z-Coordinate

Coordinate data for the Z-axis.

G1 Z-.125

Special Character Code Definitions

The following is a list of commonly used special characters, their meaning, use, and restrictions.

% Program Start or End

All programs begin and end with % on a block by itself. This code is called tape rewind character (a holdover from the days when programs were loaded using paper tapes).

() Comments

Comments to the operator must be all caps and enclosed within brackets. The maximum length of a comment is 40 characters and all characters are capitalized.

(T2: .375 END MILL)

/ Block Delete

Codes after this character are ignored if the Block Delete switch on the control is on.

/M0

; **End of Block**

This character is not visible when the CNC program is read in a text editor (carriage return), but does appear at the end of every block of code when the program is displayed on the machine control.

N8 Z0.1 ;

5.4 G&M Codes

G&M Codes make up the most of the contents of the CNC program. The definition of each class of code and specific meanings of the most important codes are covered next.

G-Codes

Codes that begin with G are called preparatory words because they prepare the machine for a certain type of motion. The most common G-codes are shown in Table 1 and a complete list and their meaning is included in Appendix B, G-M Code Reference.

Note: Older controls required G-codes to be written with a leading zero, like G01. Most machines now allow the leading zero to be omitted, as shown in the table below.

Code	Meaning
G0	Rapid motion. Used to position the machine for non-milling moves.
G1	Line motion at a specified feed rate.
G2	Clockwise arc.
G3	Counterclockwise arc.
G4	Dwell.
G28	Return to machine home position.
G40	Cutter Diameter Compensation (CDC) off.
G41	Cutter Diameter Compensation (CDC) left.
G42	Cutter Diameter Compensation (CDC) right.
G43	Tool length offset (TLO).
G54	Fixture Offset #1.
G55	Fixture Offset #2.
G56	Fixture Offset #3.
G57	Fixture Offset #4.
G58	Fixture Offset #5.
G59	Fixture Offset #6.
G80	Cancel drill cycle.
G81	Simple drill cycle.
G82	Simple drill cycle with dwell.
G83	Peck drill cycle.
G84	Tap cycle.
G90	Absolute coordinate programming mode.
G91	Incremental coordinate programming mode.
G98	Drill cycle return to Initial point (R).
G99	Drill cycle return to Reference plane (last Z Height)

Table 1: Common G-Codes

M-Codes

Codes that begin with M are called miscellaneous words. They control machine auxiliary options like coolant and spindle direction. Only one M-code can appear in each block of code.

The table below lists the most common M codes and their meaning. A complete list of M-codes is included in Appendix B, G-M Code Reference.

Code	Meaning
M0	Program stop. Press Cycle Start button to continue.
M1	Optional stop. Only executed if Op Stop switch on the CNC control is turned ON.
M2	End of program.
M3	Spindle on Clockwise.
M4	Spindle on Counterclockwise.
M5	Spindle stop.
M6	Change tool.
M8	Coolant on.
M9	Coolant off.
M30	End program and press Cycle Start to run it again.

Table 2: Common M-Codes

5.5 Select G-Code Definitions (Expanded)

G0 Rapid Move

This code commands the machine to move as fast as it can to a specified point. It is always used with a coordinate position and is modal. Unlike G1, G0 does not coordinate the axes to move in a straight line. Rather, each axis moves at its maximum speed until it is satisfied. This results in “dogleg” motion as shown in Figure 2, below.

G0 X0. Y0.

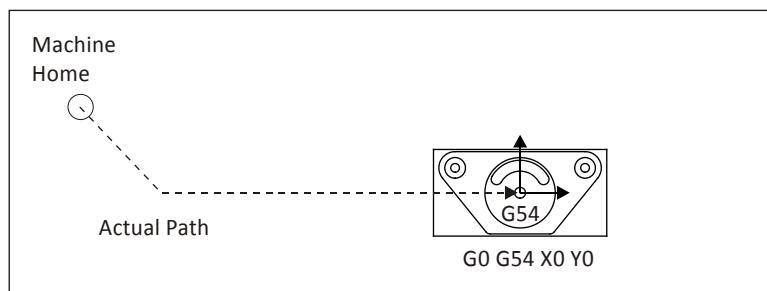


Figure 2: G0 Dogleg Motion

Caution: The rapid speed of some machines can exceed 1,000 IPM. A incorrect offset or coordinate move can crash the machine faster than the operator can hit the emergency stop. Use the rapid feed override on the machine when running a program for the first time.

G1 Linear Move

This command moves the tool in a straight line at a programmed feed rate.

G1 X1. Y1.1255 F32.

G2/G3 CW/CCW Arc

G2 commands clockwise arcs. G3 commands counterclockwise arcs. Arcs must exist on a plane (G17/G18/G19) and include the coordinates of the arc end point and IJK vectors indicating the arc center location.

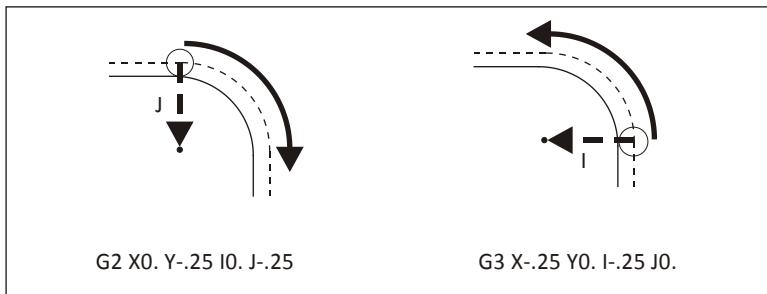


Figure 3: G2/G3 Arcs

G17/G18/G19 Plane Designation

Arcs must exist on a plane designated by the command G17 (XY), G18 (XZ) or G19 (YZ). G17 is the machine default.

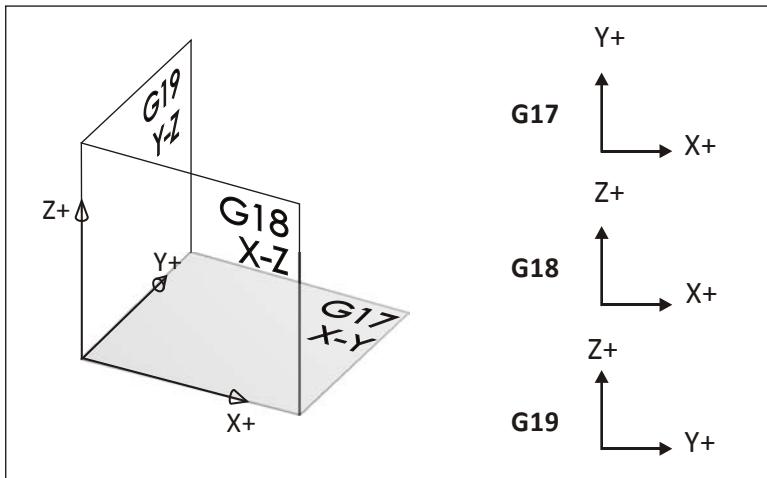


Figure 4: Plane Designations

G40/G41/G42 Cutter Diameter Compensation (CDC)

CDC is a key to precision CNC machining, allowing the operator to compensate for tool wear and deflection by commanding the machine to veer left (G41) or right (G42) from the programmed path. G40 cancels cutter compensation. The amount of offset is entered in a CNC control D-register. The wear register can be thought of like a table that the control refers to with every move.

Tool	Diameter Offset	Value
D1		0.0020
D2		0.0000
D3		0.0000
D4		0.0000
D5		0.0000
D6		0.0000

Figure 5: Diameter Offset Register

The value in the D-register is calculated by the machine tool operator, who monitors the finished size of part features, compares them with the print, and enters the difference in the register as needed to keep the part within specifications. If there is no deviation, the register is set to zero.

G1 G41 D1 X1.0 Y.25 F36.

G43 Tool Length Compensation

G43 activates tool length compensation. It is always accompanied by an H-code and Z-move, where H is the tool length offset (TLO) register to read, and Z is the height to go to in reference to the part datum.

The (TLO) can be thought of like a table on the control:

Tool Length Register	Z
H1	12.6280
H2	6.3582
H3	9.7852
H4	6.8943
H5	10.5673
H6	7.1258

Figure 6: Work Offsets

The TLO is combined with the active fixture offset on the control so the machine knows where the tip of the tool is in relation to the part datum. The process for finding the TLO detailed in Lesson 6, CNC Operation.

G43 H1 Z1.

G54 Work Offset

Work offsets are data registers in the CNC control that hold the distance from the machine home X, Y, Z position to the part datum. These offsets can be thought of like a table on the control:

Work Offset	X	Y	Z
G54	14.2567	6.6597	2.0183
G55	0.0000	0.0000	0.0000
G56	0.0000	0.0000	0.0000
G57	0.0000	0.0000	0.0000
G58	0.0000	0.0000	0.0000
G59	0.0000	0.0000	0.0000

Figure 7: Work Offsets

Tip: G54 is usually used for the first machining setup. Additional offsets are used to machine other sides of the part.

The X and Y values represent the distance from the machine home to part datum XY. The Z value is the distance from the tool reference point (for example, the top of a 1-2-3 block) and the part Z-datum. The process for finding TLO and fixture offset Z is detailed in Lesson 6, CNC Operation.

G54 X0. Y0.

5.6 Canned Cycles

Canned cycles are special codes that act like a macro. They are used for hole making and allow one compact block of code to command many moves. For example, a hole can be created using a peck drill cycle with two lines of code (left column) whereas the same move would require maybe twenty or more lines of code if each motion was commanded separately (right column).

Canned Cycle	Equivalent Motion: Expanded Code
N70 G98 G83 X1. Y1. Z-1.04 R0.06 Q0.15 P0 F9. N75 G80	N70 Z0.06 N75 Z0.04 N80 G01 Z-0.19 F9. N85 G00 Z0.06 N90 Z-0.11 N95 G01 Z-0.34 N100 G00 Z0.06 N105 Z-0.26 N110 G01 Z-0.49. N115 G00 Z0.06 N120 Z-0.41 N125 G01 Z-0.64. N130 G00 Z0.06 N135 Z-0.56 N140 G01 Z-0.79 N145 G00 Z0.06 N150 Z-0.71 N155 G01 Z-0.94. N160 G00 Z0.06 N165 Z-0.86 N170 G01 Z-1.04. N175 G00 Z0.25

Figure 8: Canned Cycle vs. Expanded Code

G81 Simple Drill Cycle

This cycle makes holes by feeding to depth at a programmed feed rate and then retracting at rapid rate. It is accompanied by G98 or G99, XYZ coordinates, feed rate, and R. R is the feed plane and Z is final depth of the tool tip.

All drill cycles are accompanied by G98 or G99 that determine how high the tool retracts between holes.

G0 Z1. G43 H1

G98 G81 X.5 Y.5 Z-1. R.1 F9.5

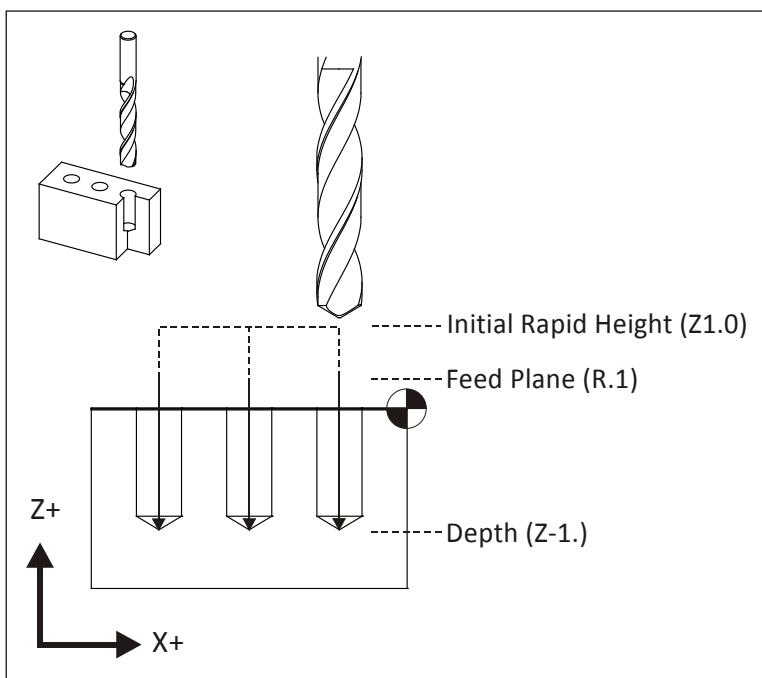


Figure 9: G81 Simple Drill Cycle

G82 Spot Drill Cycle

This cycle is identical to G81 except it includes a dwell value, P (in seconds). P is used to pause the tool feed rate at the final depth to create a clean countersink or counterbore finish.

G0 Z1. G43 H1

G98 G82 X.5 Y.5 Z-.0925 P.1 R0.1 F9.5

G83 Peck Drill

A peck drill cycle is used on deep holes. The tool drills an incremental distance (Q) and then fully retracts from the hole. This breaks the chip, clears material out of the hole, and allows coolant to cool the drill and flush out the hole, reducing the chance of the tool breaking and producing a better quality hole. The simplest form of this cycle is shown in Figure 8. Another version of this cycle, called a "deep drill cycle", uses I,J,K parameters to reduce the amount of peck as the hole gets deeper.

G0 Z1. G43 H1

G83 X.5 Y.5 Z-1. R0.1 Q.25 F9.

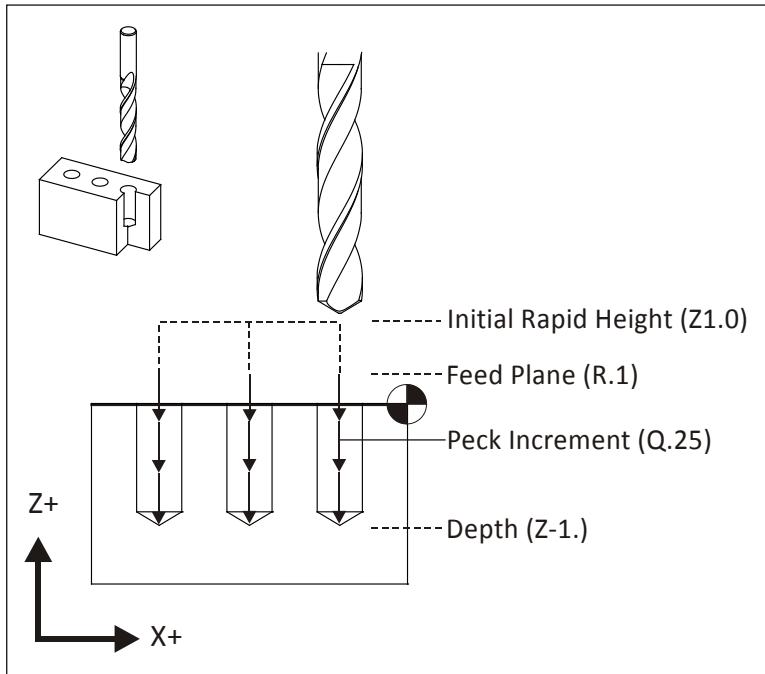


Figure 10: G83 Peck Drill Cycle

G84 Tap Cycle

Most modern machines support rigid tapping, which eliminates the need to use special tapping attachments. Rigid tapping precisely coordinates the spindle speed and feed to match the lead of the thread. It then stops and reverses the spindle at the bottom of the cycle to retract the tap. The parameters for the tap cycle are identical to simple drilling (G81).

G0 Z1. G43 H1

G84 X.5 Y.5 Z-1.5 R0.1 F20.

G90 Absolute Positioning

This code commands the machine to interpret coordinates as absolute position moves in the active Work Coordinate System. All programs are written in absolute coordinates.

G90 G0 X1. Y1.

G91 Incremental Positioning

This code commands the machine to interpret coordinates as incremental position moves. G91 is used by subprograms but most programming done with CAD/CAM software and does not use subprograms.

The only common use of G91 is in combination with G28 to send the machine back to its home position at the end of the program. The machine must be set back to G90 mode in the next block as a safety measure.

G91 G28 Z0.

G90

G98 Return to Initial Rapid Height

This code is used in drill cycles to retract the tool to the clearance plane (set in the next previous block) between holes to avoid clamps.

G0 Z1. G43 H1

G98 G81 Z-0.325 R0.1 F12.

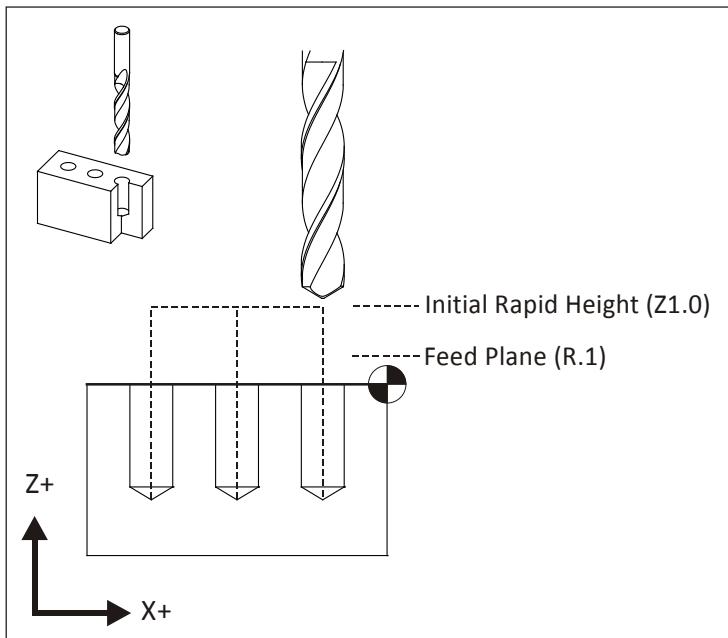


Figure 11: G98 (Return to Clearance Plane)

G99 Return to R-Plane

This code is used in drill cycles to retract the tool to the rapid plane (R) between holes. G99 mode is the machine default and is used when clamp clearance between holes is not an issue.

G0 Z1. G43 H1

G99 G81 Z-0.325 R0.1 F12.

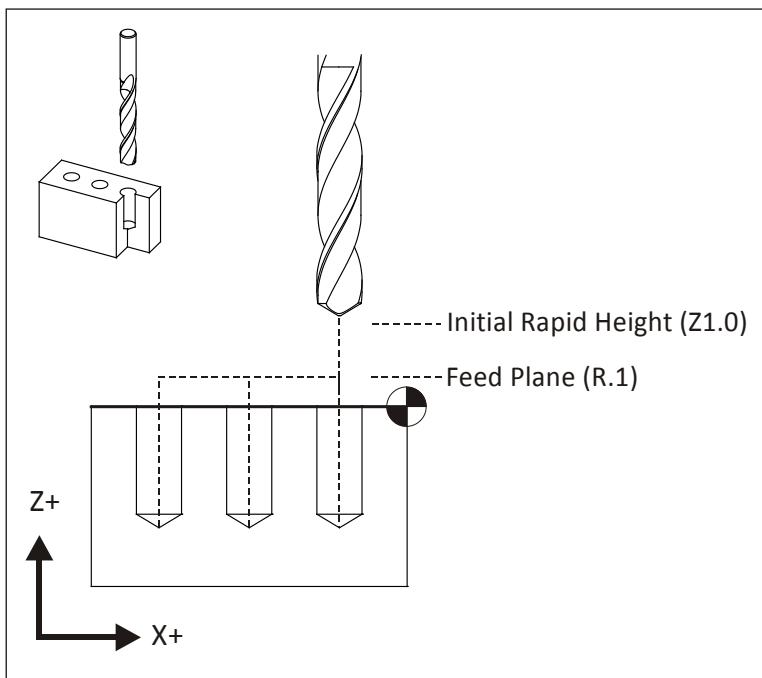


Figure 12: G99 Motion (Return to R-Plane)

Lesson 6

CNC Operation

Upon successful completion of this lesson, you will be able to:

- List the steps to set up and operate a CNC mill.
- Identify the location and purpose of the operating controls on the Haas CNC Mill control.
- Start and home a CNC machine.
- Load tools into tool carousel.
- Set Tool Length Offsets.
- Set Fixture Offsets.
- Load a CNC program into the machine control.
- Safely run a new CNC program.
- Adjust offsets to account for tool wear and deflection.
- Shut down a CNC machine correctly.

6.1 Overview of CNC Setup and Operation

CNC machine setup and operation follows the process shown in Figure 1:

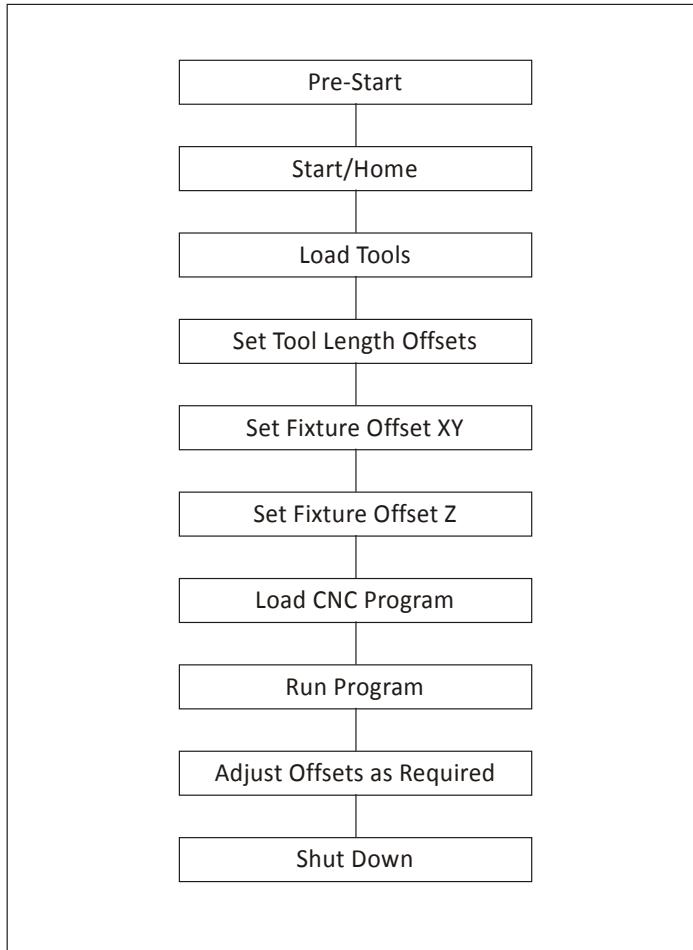


Figure 1: CNC Process

1 Pre-Start

Before starting the machine, check to ensure oil and coolant levels are full. Check the machine maintenance manual if you are unsure about how to service it. Ensure the work area is clear of any loose tools or equipment. If the machine requires an air supply, ensure the compressor is on and pressure meets the machine requirements.

2 Start/Home

Turn power on the machine and control. The main breaker is located at the back of the machine. The machine power button is located in the upper-left corner on the control face.

3 Load Tools

Load tools into the tool carousel in the order listed in the CNC program tool list.

4 Set Tool Length Offsets

For each tool used, jog the machine to find and then set the TLO.

5 Set Fixture Offset XY

Once the vise or other fixture is properly installed and aligned on the machine, set the fixture offset to locate the part XY datum.

6 Set Fixture Offset Z

Use a dial indicator and 1-2-3 block to find and set the fixture offset Z.

7 Load CNC Program

Download the CNC program from your computer to the machine control using RS-232 communications, USB flash memory, or floppy disk.

8 Run Program

Run the program, using extra caution until the program is proven to be error-free.

9 Adjust Offsets as Required

Check the part features and adjust the CDC or TLO registers as needed to ensure the part is within design specifications.

10 Shut Down

Remove tools from the spindle, clean the work area, and properly shut down the machine. Be sure to clean the work area and leave the machine and tools in the location and condition you found them.

Warning

Never operate a CNC machine or any shop equipment unless you have been properly trained on its use.

Observe extreme caution at all times.

Follow all safety rules.

6.2 Machine and Tool Offsets

Fixture Offset XY

Figure 2 shows a plan view of how the Fixture Offset XY works. The CNC operator finds the fixture offset values by jogging (moving) the machine from the machine at its home position the CNC program datum. This can be any point on the part, stock, or fixture, as long as it can be found by mechanical means such as an edge finder or part probe.

The incremental X and Y distances moved between points is recorded and entered into a **Fixture Offset Register** on the CNC control. Think of the Offset registers like a table in a spreadsheet. The CNC control references these values in the table each time a motion is commanded, adding or subtracting them from coordinates in the CNC program. In other words, Fixture XY offsets convert Machine Coordinates into WCS coordinates.

Most machine controls support at least six fixture offsets, labeled G54 thru G59. Multiple registers are needed because most parts use a different fixture offset for each side of the part machined.

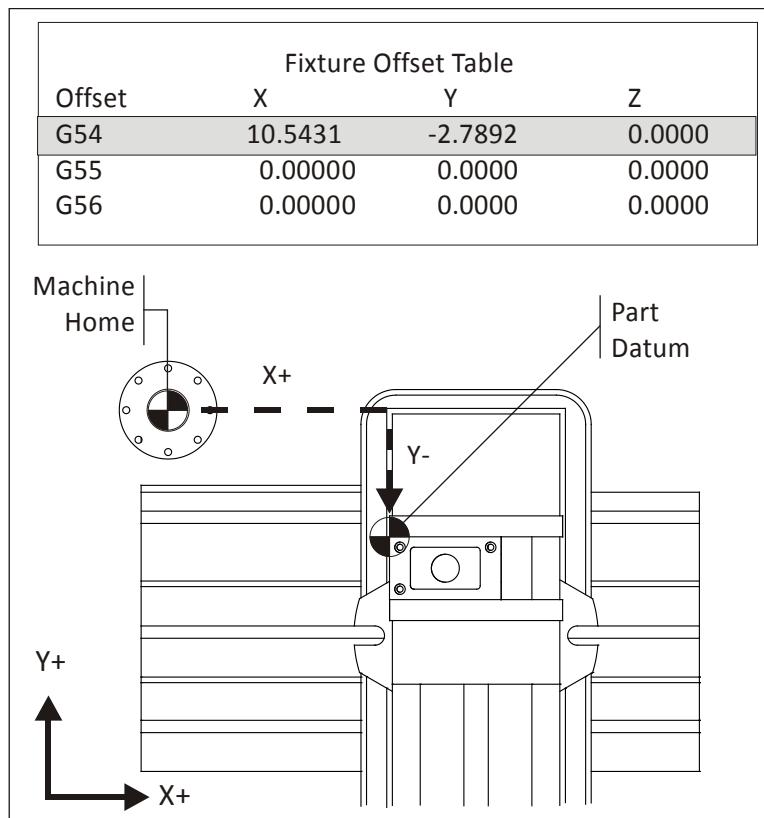


Figure 3: Fixture Offset Plan View

The Fixture Offset Z-coordinate is not used to shift the Machine Z. Use of the Fixture Offset Z is covered in the next topic.

Fixture Offset Z

The purpose of the Fixture Offset Z is to record the incremental distance from a tool setting position to the part datum. The tool set position can be a tool probe or, as shown in Figure 10, the top of a precision 1-2-3 block set on the machine table. The approach shown in Figure 10 involves using a dial indicator and this process is detailed in Lesson 6 (CNC Operation: Set Fixture Offset Z).

The CNC control adds the Fixture Offset Z and Tool Length Offset for the active tool together to calculate the distance from the tip of each tool at Home to the Z-datum on the part.

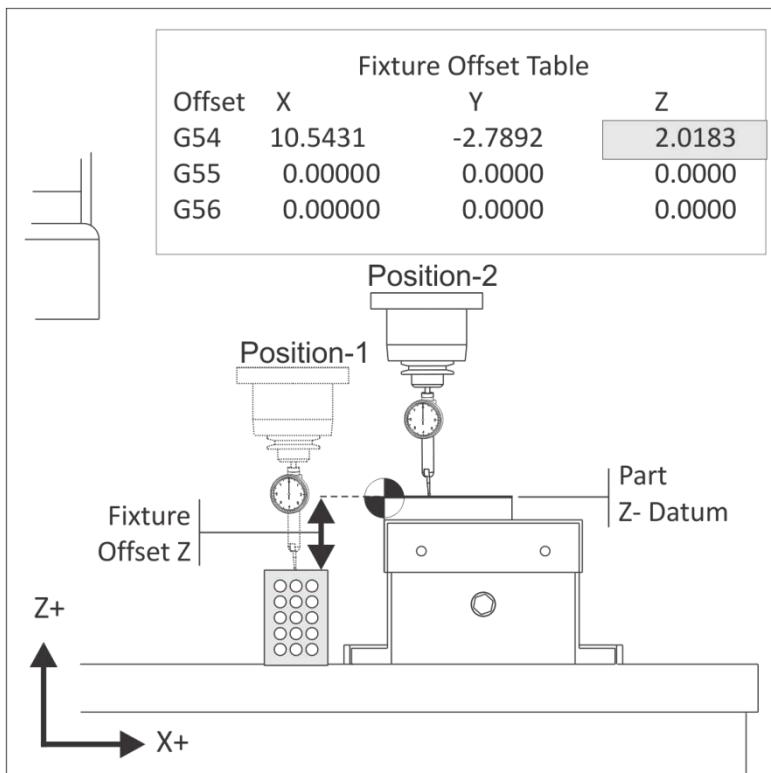


Figure 10: Fixture Offset Z

There are many ways to set tool and fixture offsets. The method described here and detailed in *Lesson 6, CNC Operation*, is precise, compatible with tool probe systems, and easy to understand and use once tried just a few times. Another advantage is that the TLO can be reset easily, even if the part datum has been machined away, which is common with 3D parts.

Appendix-E: Alternate Tool Setting Methods describes three other methods that can be used to set up the machine tool length offsets.

To use the method described in this lesson on Haas Automation machines, machine control parameter 64 (T OFS USES WORK) must be set to OFF. Refer to the Haas Programming and Operation manual for instructions on set this parameter.

Tool Length Offset (TLO)

Every tool loaded into the machine is a different length. In fact, if a tool is replaced due to wear or breaking, the length of its replacement will likely change because it is almost impossible to set a new tool in the holder in exactly the same place as the old one. The CNC machine needs some way of knowing how far each tool extends from the spindle to the tip. This is accomplished using a **Tool Length Offset (TLO)**.

The TLO is found by jogging the spindle with tool from the machine home Z-position to the tool setting point on the machine. This can be the top of a tool probe, or as shown in Figure 11, the top of a 1-2-3 block resting on the machine table. The distance travelled from home to the top of the block is recorded, and the value entered into the TLO register for that tool (called an H-register, because it is preceded by the letter H in the CNC program).

If a tool wears or breaks, it can be replaced, the H-register reset to the new tool by touching off again on the 1-2-3 block, and the program continued with no other changes.

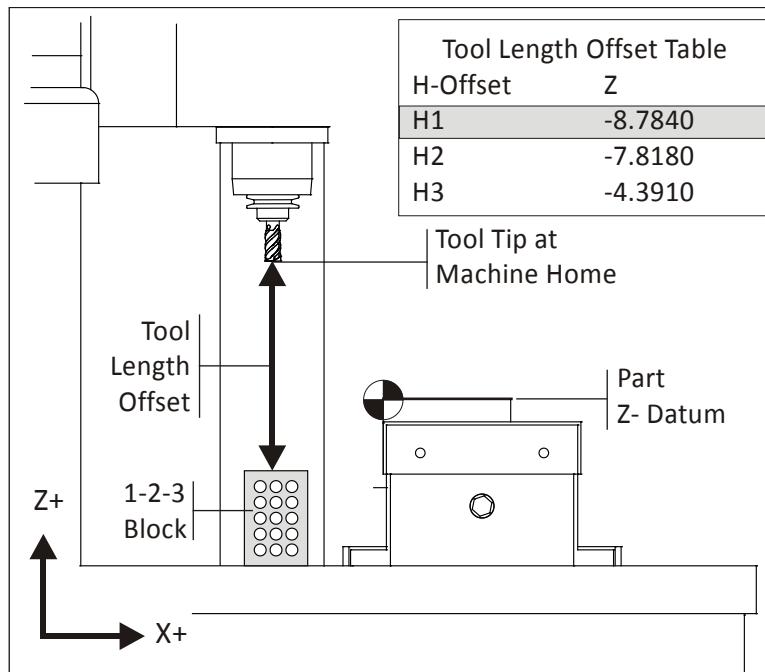


Figure 11: Tool Length Offset

Haas Control

The Haas control is shown in Figures 2 and 3. Familiarize yourself with the location of buttons and controls. Detailed instructions on the following pages show how to operate the control.

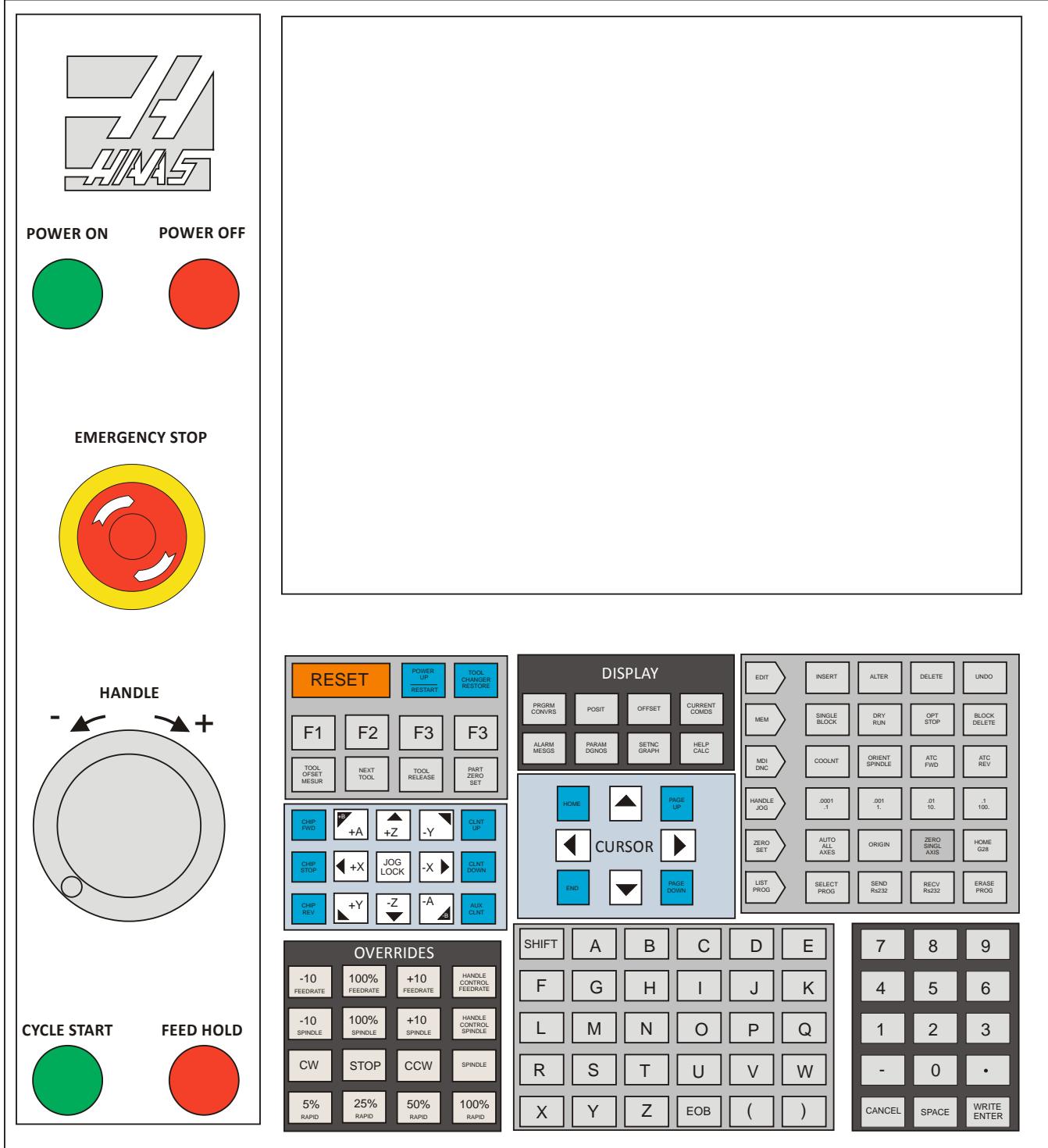


Figure 2: Haas CNC Mill Control

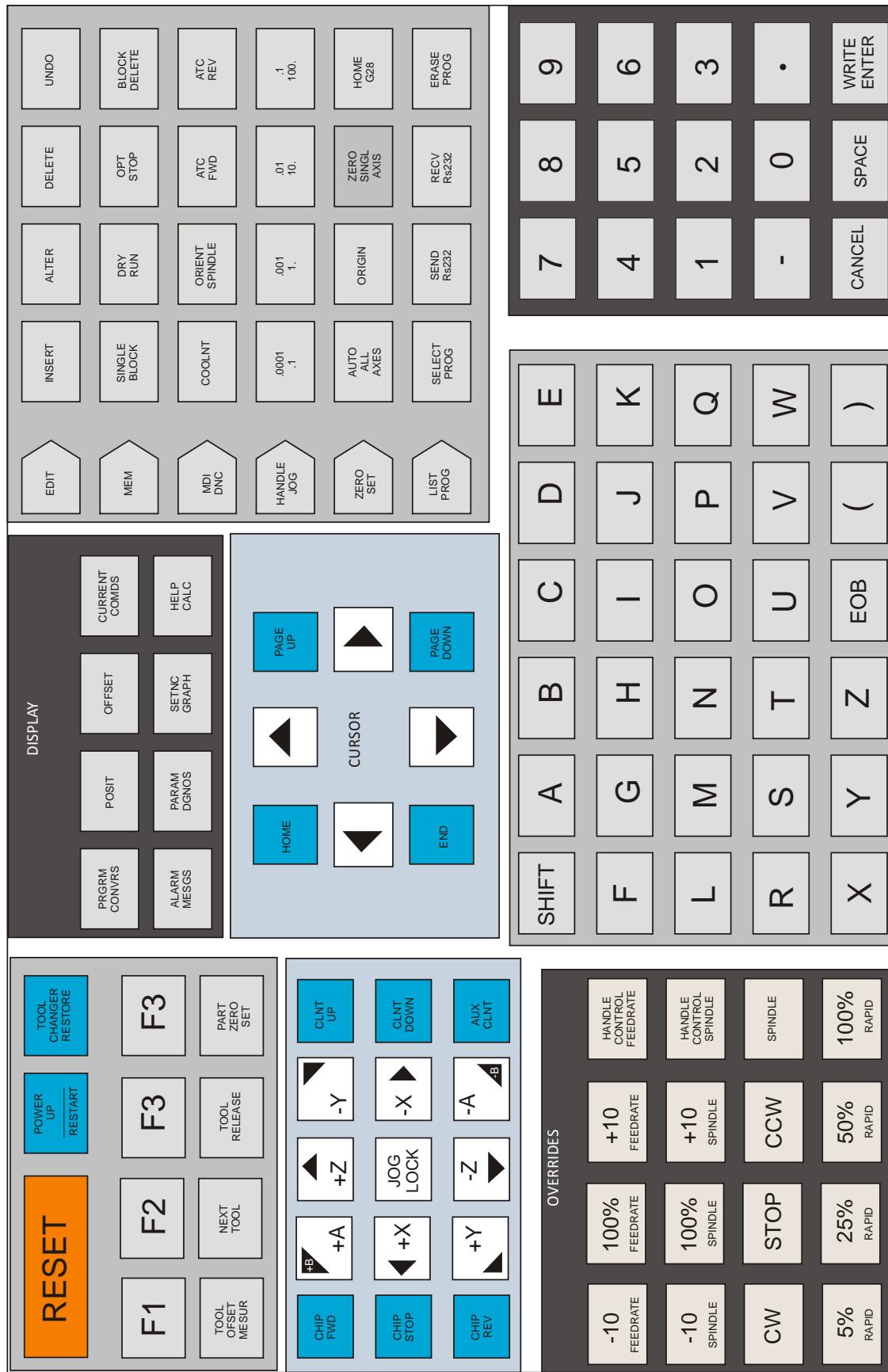
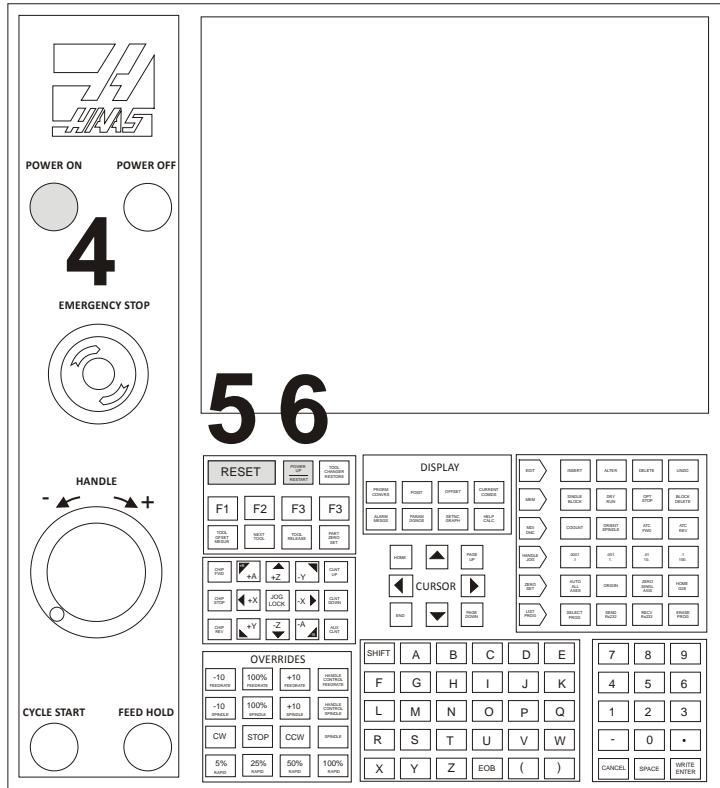


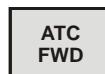
Figure 3: Haas CNC Control Buttons

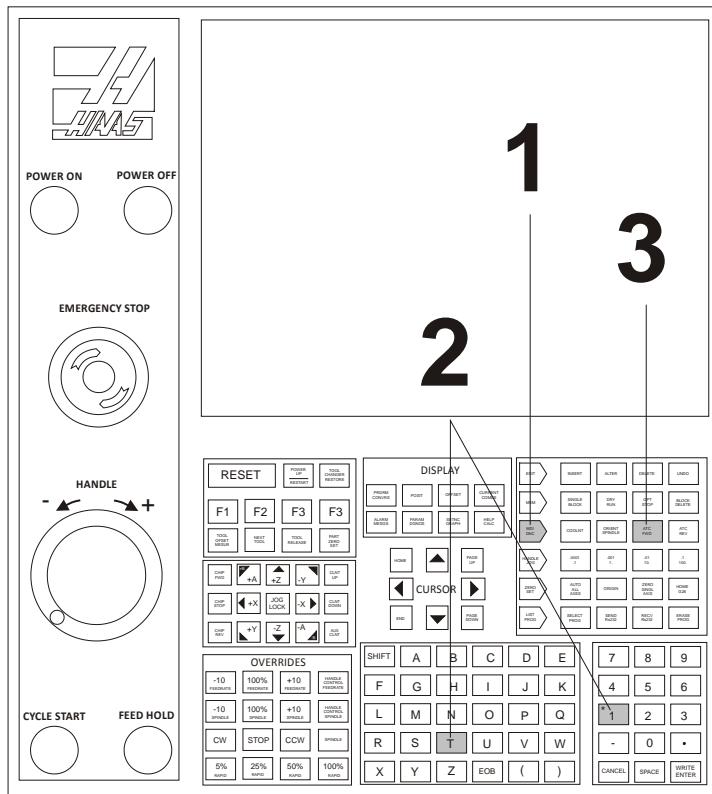
Start/Home Machine

Check List		Buttons
1	Work Area: Clear	
2	Air Supply: On/Correct <ul style="list-style-type: none"> ▪ Haas requires at least 70PSI for tool changer to operate. 	
3	Main Breaker: On	
4	POWER ON: Press <ul style="list-style-type: none"> ▪ Ensure Emergency Stop is not tripped. If it is, twist red knob right to release. ▪ Wait until message 102 SERVOS OFF appears before proceeding. 	<p style="text-align: center;">POWER ON</p> 
5	RESET: Press	
6	Power On Restart: Press <ul style="list-style-type: none"> ▪ Ensure doors are closed and work area is clear. ▪ Allow all machine axes to home before proceeding. 	



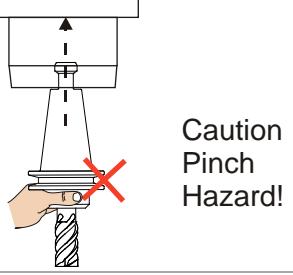
Load Tools

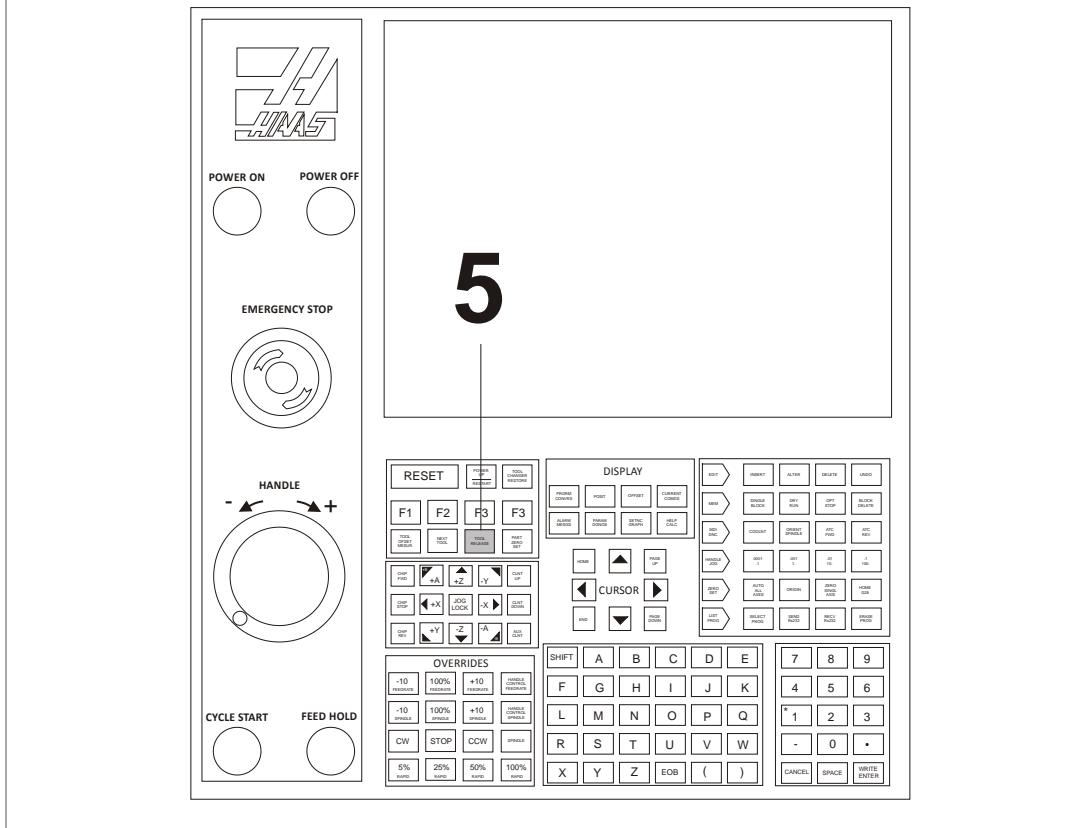
Check List		Buttons
1	MDI/DNC Key: Select	
2	Tool Number: Enter <ul style="list-style-type: none"> ▪ For example, to position the tool changer to T1, Press the T and then the 1 buttons. 	
3	ATC FWD: Select <ul style="list-style-type: none"> ▪ Tool carousel will index to T1 position. 	



Load Tool

Sheet 1 of 2

4	Position Tool in Spindle <ul style="list-style-type: none"> ▪ Do not grip by tool cutting flutes! ▪ Ensure tool taper is clean. ▪ Grip tool holder below V-flange to prevent pinching. ▪ Push tool into spindle. ▪ Ensure “dogs” on spindle line up with slots on tool holder. 	 <p>Caution Pinch Hazard!</p>
5	Tool Release: Select <ul style="list-style-type: none"> ▪ Machine will blow air thru spindle to clear debris. ▪ Gently push the tool upward and then release the Tool Release button. ▪ Ensure tool is securely gripped by spindle before releasing it. 	 <p>TOOL RELEASE</p>
6	Repeat <ul style="list-style-type: none"> ▪ Repeat steps 2-5 until all tools are loaded. 	

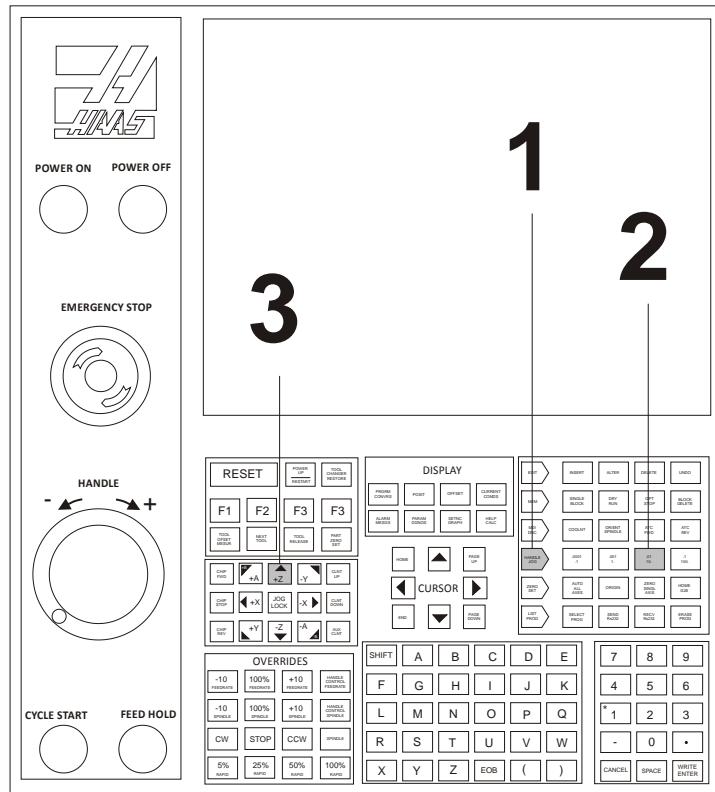


Load Tool

Sheet 2 of 2

Set Tool Length Offset (TLO)

Check List		Buttons
1	Handle Jog Mode: Select <ul style="list-style-type: none"> This sets machine to be controlled by the hand wheel. 	
2	Jog Increment: .01 <ul style="list-style-type: none"> This sets the job increment so each click of the hand wheel moves the tool .01 inches in the jog direction. 	
3	Jog Direction: Z <ul style="list-style-type: none"> This sets the tool to move in Z when the jog handle is moved. 	

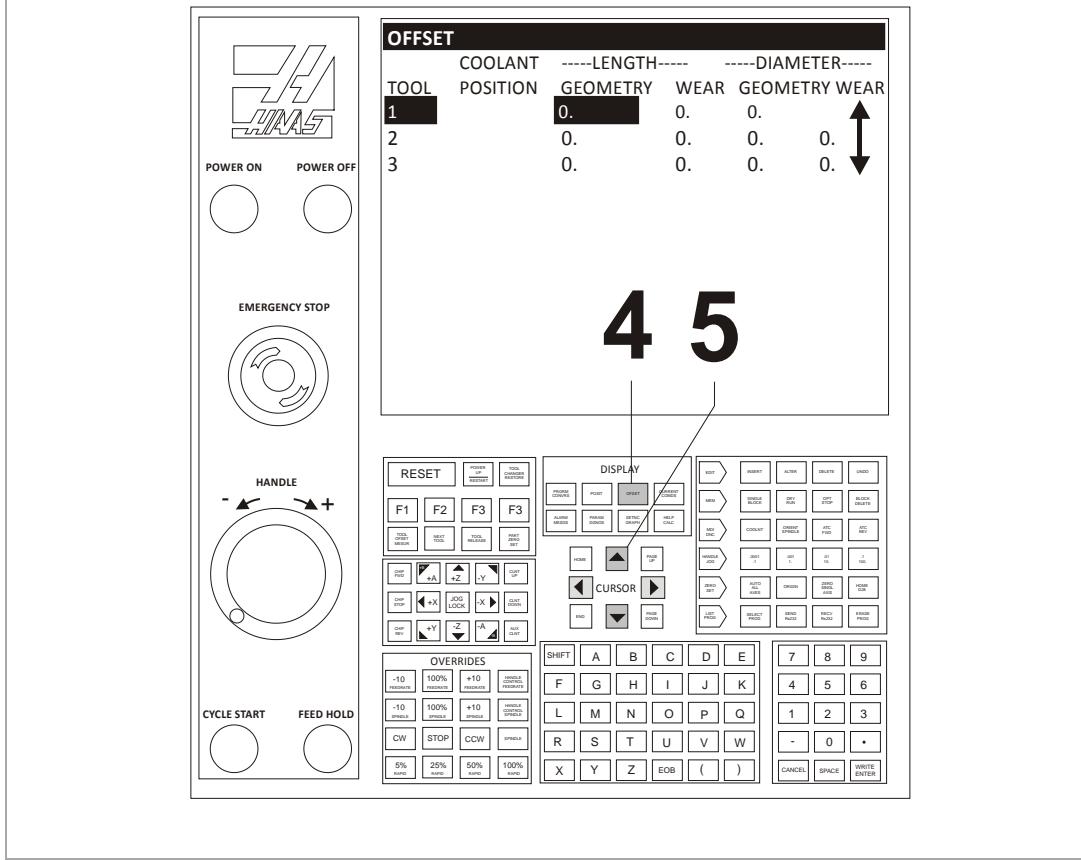


Note: Machine parameter 64 (T OFS MEAS USES WORK) must be set to OFF to use this method.

Set TLO

Sheet 1 of 4

4	Offsets: Select <ul style="list-style-type: none">▪ Press this button until the Tool Offset page displays.	
5	Cursor Arrows: Align for active tool <ul style="list-style-type: none">▪ Use the Up-Dn cursor keys (if needed) to move the highlighted bar on the graphics display over the offset values for the currently active tool.	 CURSOR 



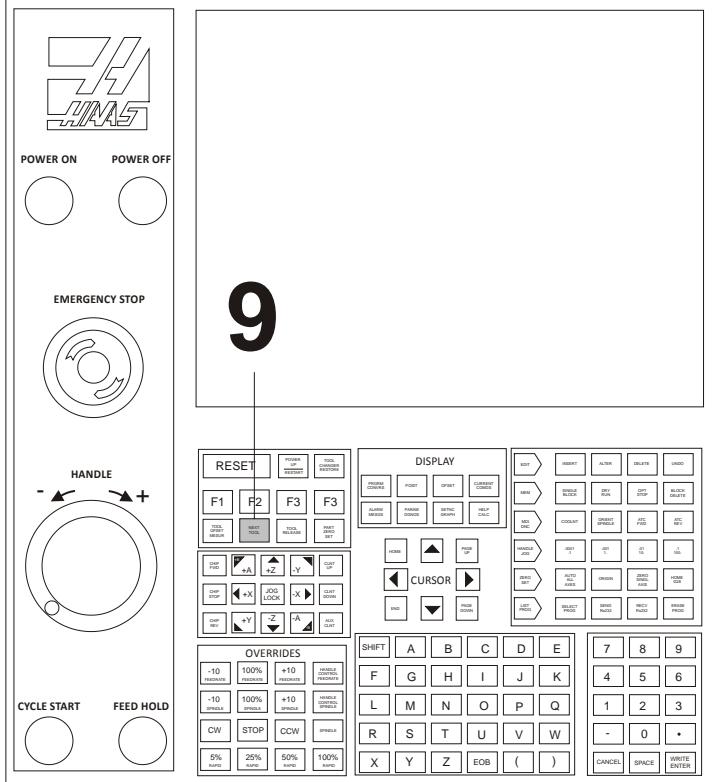
Set TLO

Sheet 2 of 4

<p>6 Use 1-2-3 Block to Set Tool Length</p> <ul style="list-style-type: none"> Jog so tool is below top block. Apply slight pressure to block against tool. Use Jog Wheel to raise tool until the block just slides underneath it. Move block out of way and then move tool back down .01 inches below top of block. 																														
<p>7 Jog Increment: .001</p> <ul style="list-style-type: none"> Reduce jog increment and use jog handle to raise tool in .001 increments until it just slides under the block again. 	<div style="text-align: center;">.001 1.</div>																													
<p>8 Tool Offset Measure: Select</p> <ul style="list-style-type: none"> This causes the control to enter the current position of the tool in the length offset register. Make sure the tool length number updates before proceeding. 	<div style="text-align: center;">TOOL OFFSET MEASUR</div>																													
<table border="1" style="margin-left: auto; margin-right: auto;"> <thead> <tr> <th rowspan="2">TOOL</th> <th rowspan="2">POSITION</th> <th>COOLANT</th> <th colspan="2">LENGTH</th> <th colspan="2">DIAMETER</th> </tr> <tr> <th>GEOMETRY</th> <th>WEAR</th> <th>GEOMETRY</th> <th>WEAR</th> </tr> </thead> <tbody> <tr> <td>1</td> <td></td> <td>-5.4203</td> <td>0.</td> <td>0.</td> <td>0.</td> </tr> <tr> <td>2</td> <td></td> <td>0.</td> <td>0.</td> <td>0.</td> <td>0.</td> </tr> <tr> <td>3</td> <td></td> <td>0.</td> <td>0.</td> <td>0.</td> <td>0.</td> </tr> </tbody> </table>	TOOL	POSITION	COOLANT	LENGTH		DIAMETER		GEOMETRY	WEAR	GEOMETRY	WEAR	1		-5.4203	0.	0.	0.	2		0.	0.	0.	0.	3		0.	0.	0.	0.	
TOOL			POSITION	COOLANT	LENGTH		DIAMETER																							
	GEOMETRY	WEAR		GEOMETRY	WEAR																									
1		-5.4203	0.	0.	0.																									
2		0.	0.	0.	0.																									
3		0.	0.	0.	0.																									

Set TLO

Sheet 3 of 4

9 Next Tool: Select <ul style="list-style-type: none"> ▪ This causes the current tool to be put away and the next tool to be loaded. ▪ Repeat steps 1 thru 9 until all tools are set. 	
	<p>9</p>

Tips

- Do not press any buttons between steps 8-9 or **NEXT TOOL** button will not load next tool.
- If you should press any buttons and interrupt this process, load the next tool by selecting the **ATC FWD** button, followed by **HANDLE JOG**.

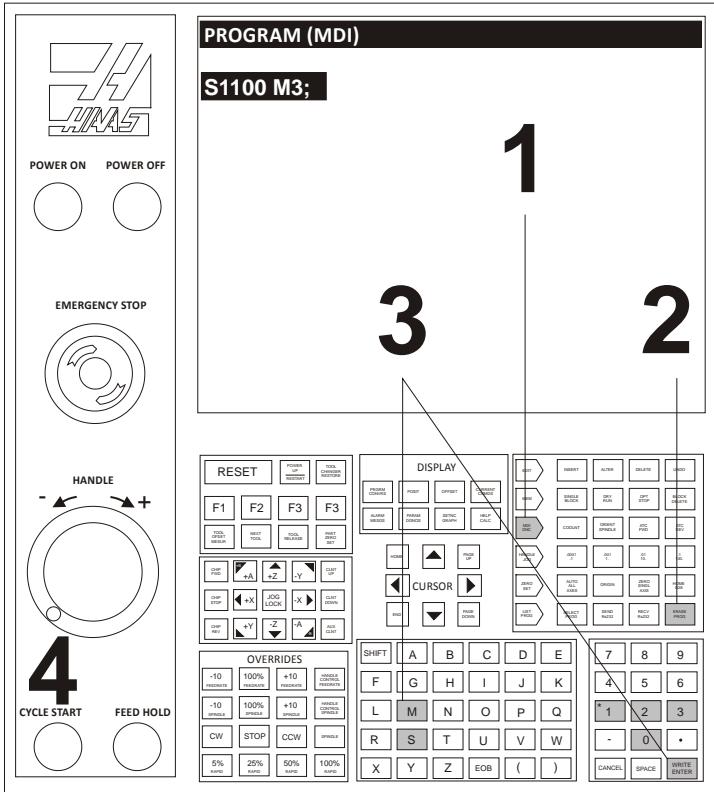
Set TLO

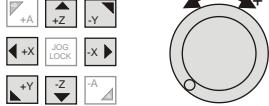
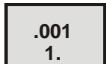
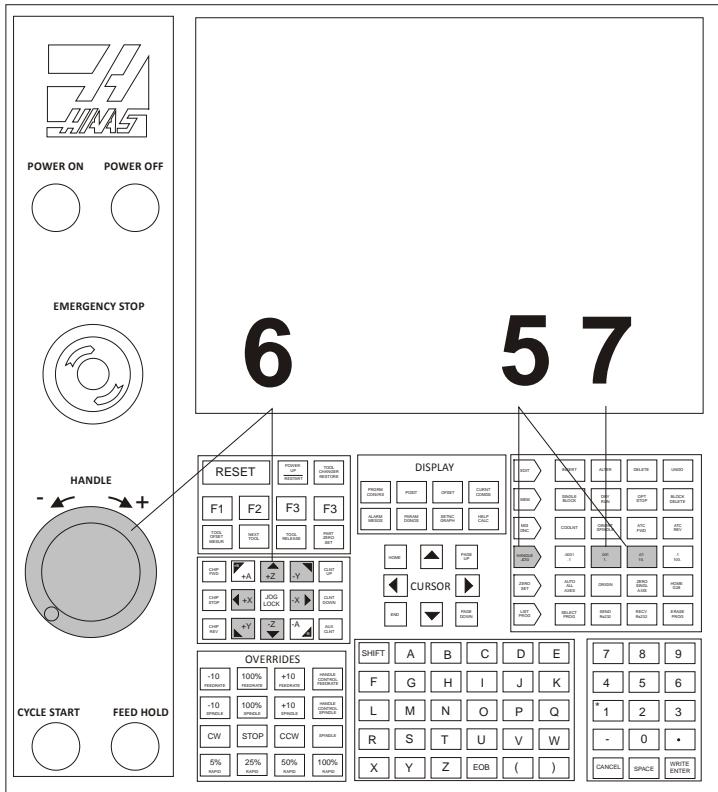
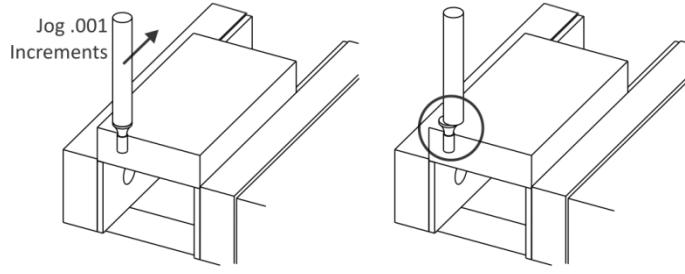
Sheet 4 of 4

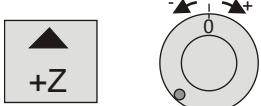
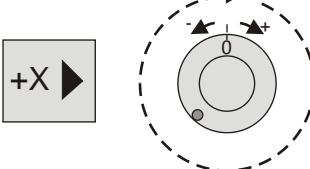
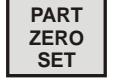
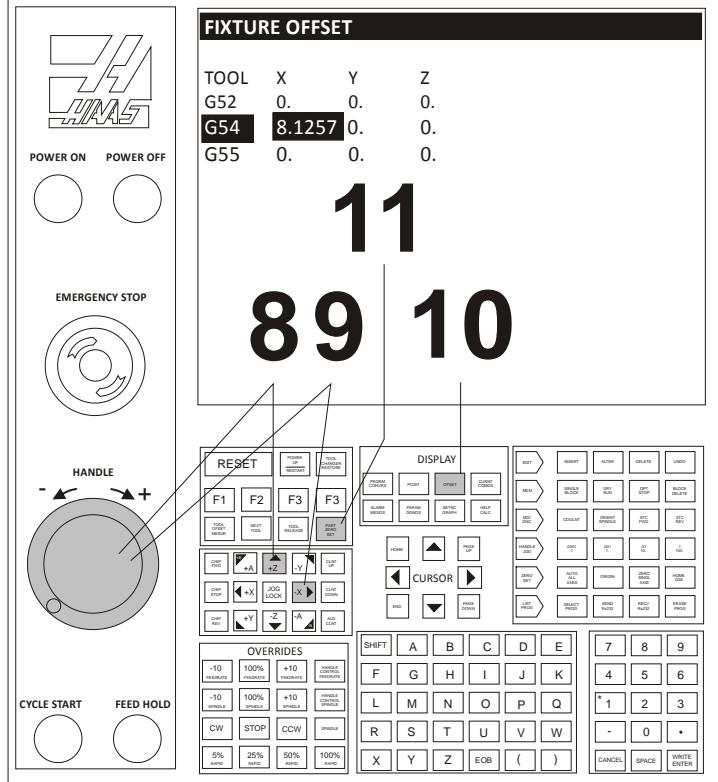
WARNING: Setting tools requires manually jogging the machine with hands in the machine work envelope. Use extreme caution and observe the following rules:

- The spindle must be off.
- Never place your hand between the tool and the 1-2-3 block.
- Ensure the correct axis and jog increment are set before jogging.
- Move the handle slowly and deliberately. Keep your eyes on your hands and the tool position at all times.
- Never allow anyone else to operate the control when your hand is in the work area.

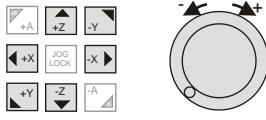
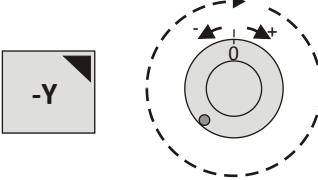
Set Fixture Offset XY

Checklist		Buttons
1	MDI/DNC Key: Select Erase Prog: Select (to clear any commands)	
2	Erase Prog: Select (to clear any commands)	
3	Spindle Speed: S1100 ▪ S1100 M3: Input ▪ Write/Enter: Select	
4	Cycle Start: Select ▪ Spindle will start CW at 1100 RPM	
 <p>The diagram illustrates the control panel setup for setting fixture offsets. On the left, a vertical control handle is labeled 'HANDLE' with arrows pointing left and right. Below it are two circular buttons labeled 'CYCLE START' and 'FEED HOLD'. Above the handle are two circular buttons labeled 'POWER ON' and 'POWER OFF'. To the right of the handle is a circular 'EMERGENCY STOP' button. The central part of the diagram shows a computer monitor displaying the 'PROGRAM (MDI)' screen. The screen shows the command 'S1100 M3;' followed by large numbers '1', '3', '2', and '1'. The number '3' is highlighted with a callout line pointing to the 'JOG LOCK' button on the control panel. The number '2' is highlighted with a callout line pointing to the 'DISPLAY' button on the control panel. The number '1' is highlighted with a callout line pointing to the 'RESET' button on the control panel. The number '1' at the bottom is highlighted with a callout line pointing to the 'WRITE ENTER' button on the control panel.</p>		

5	Handle Jog: Select Jog Increment: .01	
6	Jog Handle: As Needed <ul style="list-style-type: none">▪ Select jog direction and use handle as required to place edge finder stylus alongside the left part edge.	
7	Jog Increment: .001 <ul style="list-style-type: none">▪ Move edge finder slowly until it just trips off center as shown below.▪ This places the center of the spindle exactly .100 from the part edge.	
		
	 <p>Just Before Tripping Just After Tripping</p>	

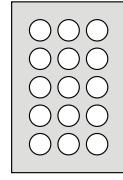
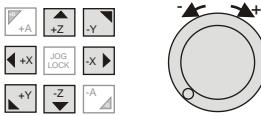
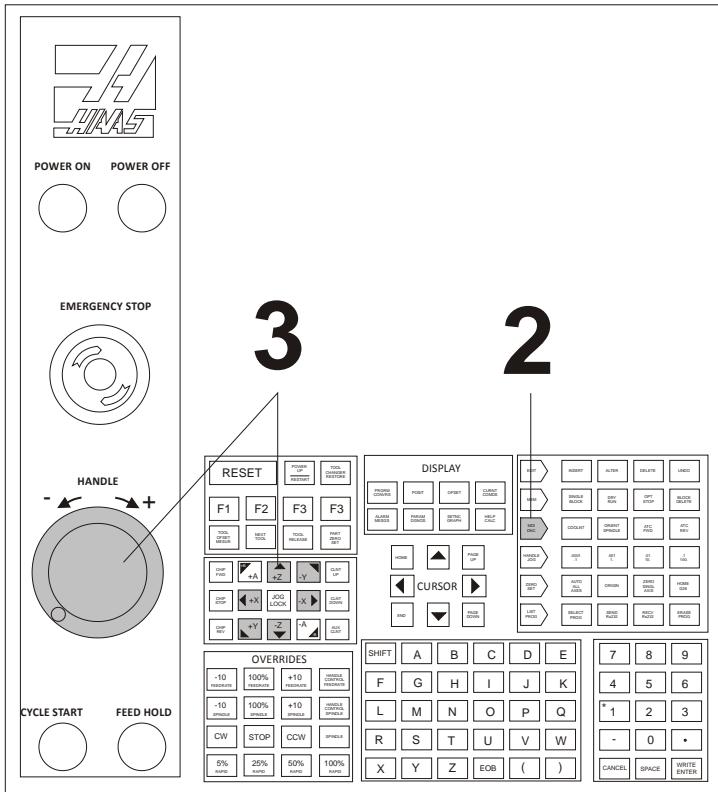
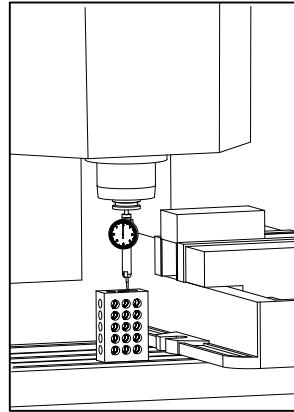
8 Jog Handle: Retract in Z <ul style="list-style-type: none"> Jog straight upward in Z until edge finder is above part and jog handle reads zero on the dial. 	
9 Jog Handle: Set jog direction to +X and rotate handle one full turn clockwise. <ul style="list-style-type: none"> Since the control is in .001 increment mode, rotating the dial exactly one full turn places the center of the spindle directly over the left part edge. 	
10 Offset Page: Select <ul style="list-style-type: none"> Select Offset button and PgUp/PgDn buttons until Work Zero Offset page appears. Use Arrow keys to highlight G54 (or whatever fixture offset is to be set). 	
11 Part Zero Set: Select <ul style="list-style-type: none"> This sets the G54 X value to the current spindle position. 	
	<p>Tips</p> <ul style="list-style-type: none"> To shift the datum RIGHT in relation to the machine operator, ADD a shift amount to the offset X-value. For example, to shift X+.1, input .1 WRITE/ENTER. To shift the datum CLOSER to the machine operator, SUBTRACT a shift amount from the offset Y-value. For example, to shift Y-.1, input -.1 WRITE/ENTER.

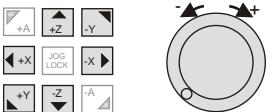
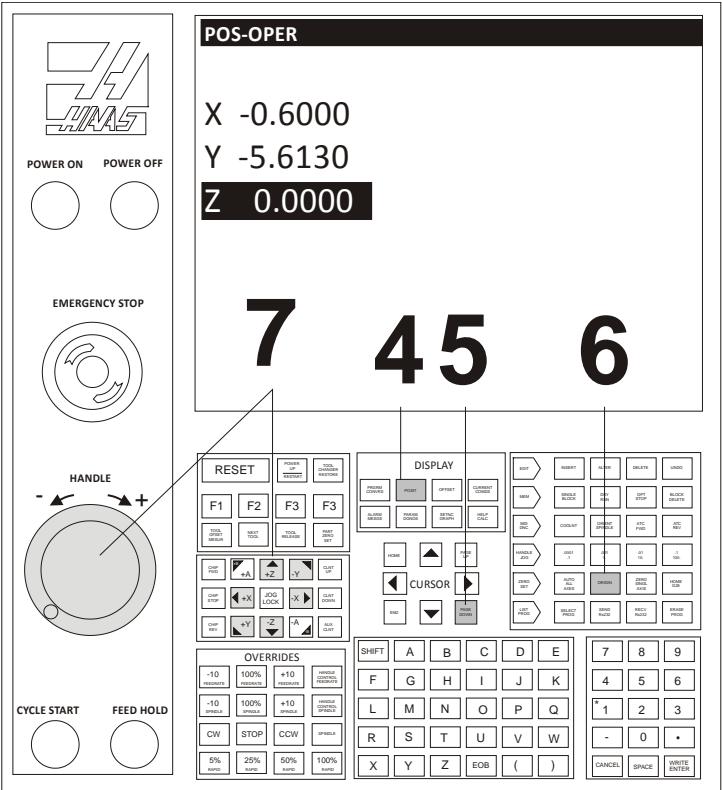
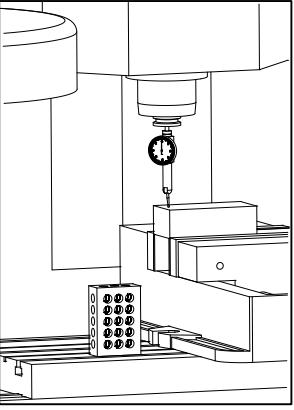
Fixture Offset XY**Sheet 3 of 4*****The following instructions repeat steps 4-10 but for setting the Y-axis.***

12	Jog Handle: As Needed <ul style="list-style-type: none"> ▪ Select jog direction and use handle as required to place edge finder stylus alongside the back part edge. 	
13	Jog Increment: .001 <ul style="list-style-type: none"> ▪ Move edge finder slowly until it just trips off center. ▪ This places the center of the spindle exactly .100 from the part edge. 	
14	Jog Handle: Retract in Z <ul style="list-style-type: none"> ▪ Jog straight upward in Z until edge finder is above part and jog handle reads zero on the dial. 	
15	Jog Handle: Set jog direction to -Y and rotate handle one full turn clockwise. <ul style="list-style-type: none"> ▪ Since the control is in .001 increment mode, rotating the dial exactly one full turn places the center of the spindle directly over the back part edge. 	
16	Offset Page: Use arrow keys to highlight G54 field (or whichever fixture offset you are setting).	
17	Part Zero Set: Select <ul style="list-style-type: none"> ▪ This sets the G54 Y value to the current spindle position. 	
18	Spindle Stop: Select	

Fixture Offset XY**Sheet 4 of 4**

Set Fixture Offset Z

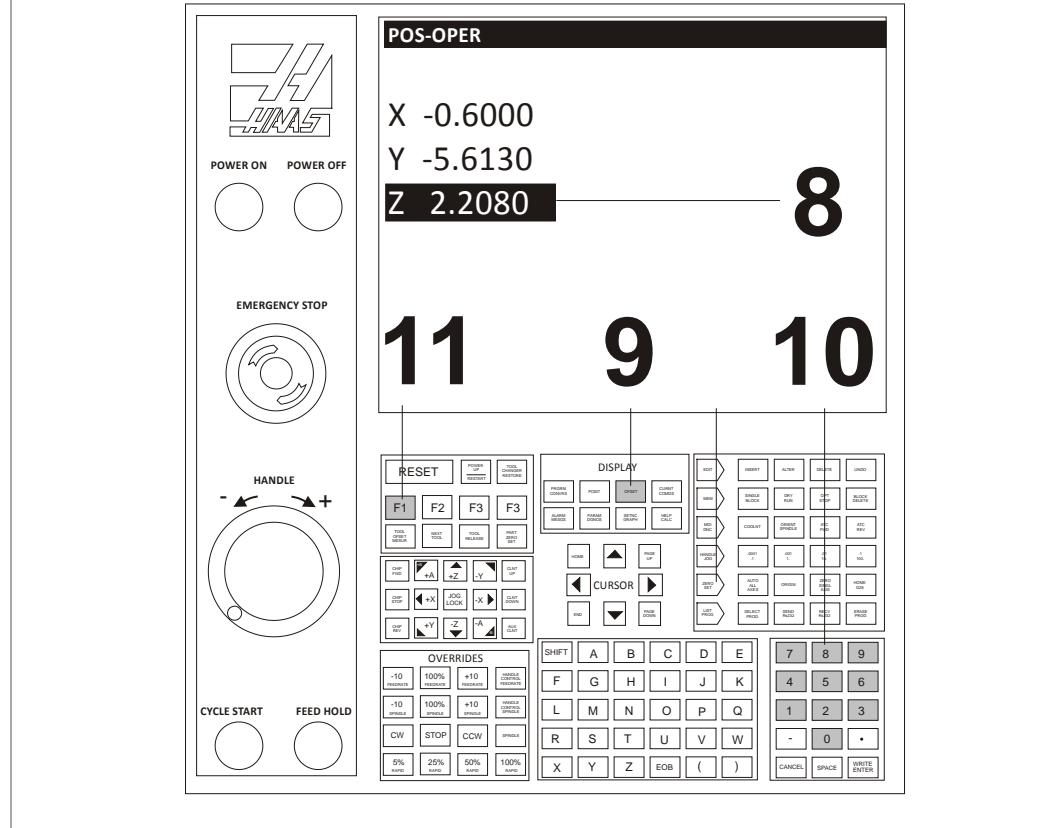
Check List		Buttons
1	Prepare: <ul style="list-style-type: none"> ▪ Ensure tip angle of dial indicator is at about 15 degrees. ▪ Ensure gauge block is flat on table, no chips underneath, and not floating on coolant. 	 Set Tip About 15 Deg From Vertical  Dial Indicator 1-2-3 Block
2	MDI Mode: Select <ul style="list-style-type: none"> ▪ Ensure machine is in MDI Mode 	
3	Handle: As Needed <ul style="list-style-type: none"> ▪ Select jog direction and use handle as required to place dial indicator stylus on top of gauge block and the dial reads zero. 	
		
<p>Note: Machine parameter 64 (T OFS MEAS USES WORK) must be set to OFF to use this method.</p>		

4	Posit Button: Press	
5	Operator Screen: Select ▪ Use PG DN Button (5 times) until POS-OPER screen appears.	
6	Origin: Set ▪ Press Origin button to set Z-value on operator screen to zero.	
7	Handle: As Needed ▪ Select jog direction and use handle as required to place dial indicator stylus on top of part stock and the dial reads zero.	
		

Fixture Offset Z

Sheet 2 of 3

8	Operator Z Value: Read <ul style="list-style-type: none">▪ Read the operator Z value from the graphic area. This is the incremental distance between the top of the 1-2-3 block and the top of the part. Ex: 2.2080	
9	Offset Page: Set G54 Z Value <ul style="list-style-type: none">▪ Press Offset button twice to get to Offsets Page	
10	Numeric Keypad: Enter Z Value <ul style="list-style-type: none">▪ Input the value using the numeric keyboard.▪ To set Z below the part face, reduce the value by the amount of material removed by the face cut.	
11	F1 Button: Press <ul style="list-style-type: none">▪ Select the F1 button to enter the value input in Step #7 into the Z offset field.	



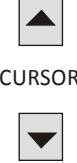
Tip

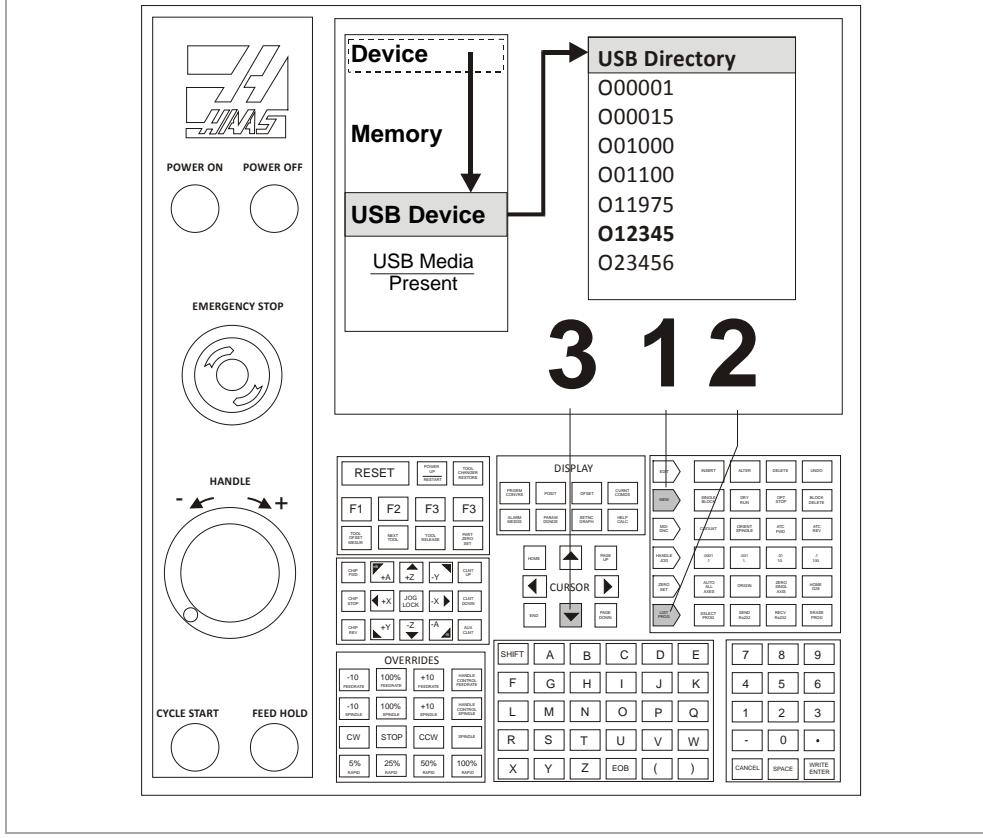
Shift the datum DOWN by an incremental distance from the top of the part to remove stock from the top-face by **subtracting** the amount of stock to remove from the Fixture Offset Z Value.

Example: To set the datum to Z-.02 below the top of the part:

$$\begin{array}{r}
 22.2080 \\
 - .020 \\
 \hline
 = 2.1880
 \end{array}$$

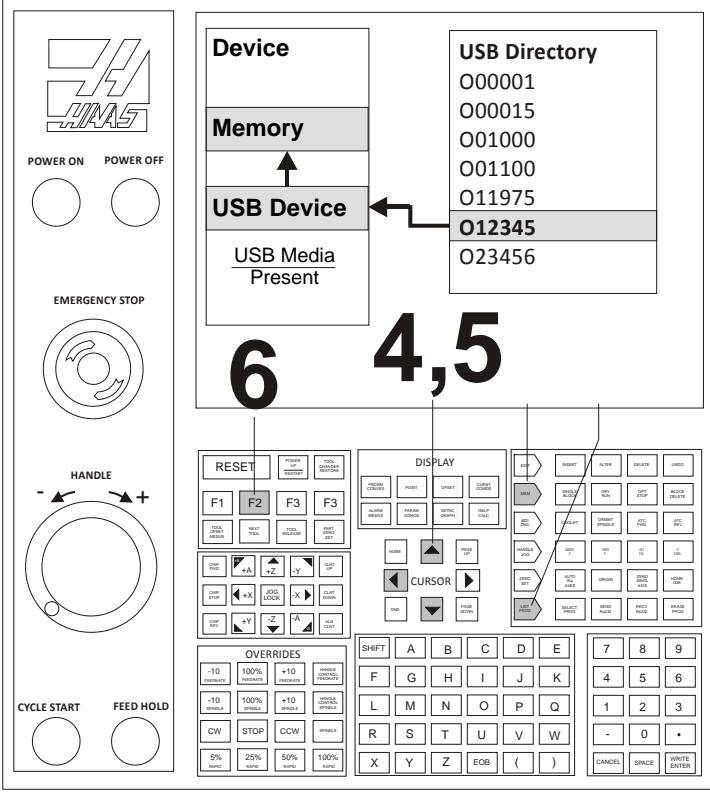
Load CNC Program

Check List		Buttons
1	Memory: Select	
2	List Programs: Select	
3	USB Device: Select <ul style="list-style-type: none"> ▪ Use the DN arrow to highlight USB Device on the graphics screen. (This sets the program list to display files on the USB Device). ▪ Select F4 button. 	 F4



Load CNC Program

Sheet 1 of 2

Checklist		Buttons
4	Program Name: Highlight <ul style="list-style-type: none"> ▪ Use RT button to move highlighted bar over USB Directory List ▪ Use the DN arrow keys to highlight the program to read. 	 CURSOR  <div style="border: 1px solid black; padding: 5px; display: inline-block;"> Write Enter </div>
5	Write To Device (Memory): Select <ul style="list-style-type: none"> ▪ Press the Left arrow key and then the Up arrow key to move the highlight bar back over the Memory option in the Device list. 	    CURSOR    
6	F2: Select <ul style="list-style-type: none"> ▪ Press the F2 button to copy the program from the USB Device to Memory. ▪ If the program already exists, select (Y) when prompted to overwrite. 	<div style="border: 1px solid black; padding: 5px; display: inline-block;"> F2 </div>
		

Note: All programs must begin with the letter "O" (NOT the number zero). The program name must be an integer up to five digits long. No decimal point, letters, or special characters are allowed.

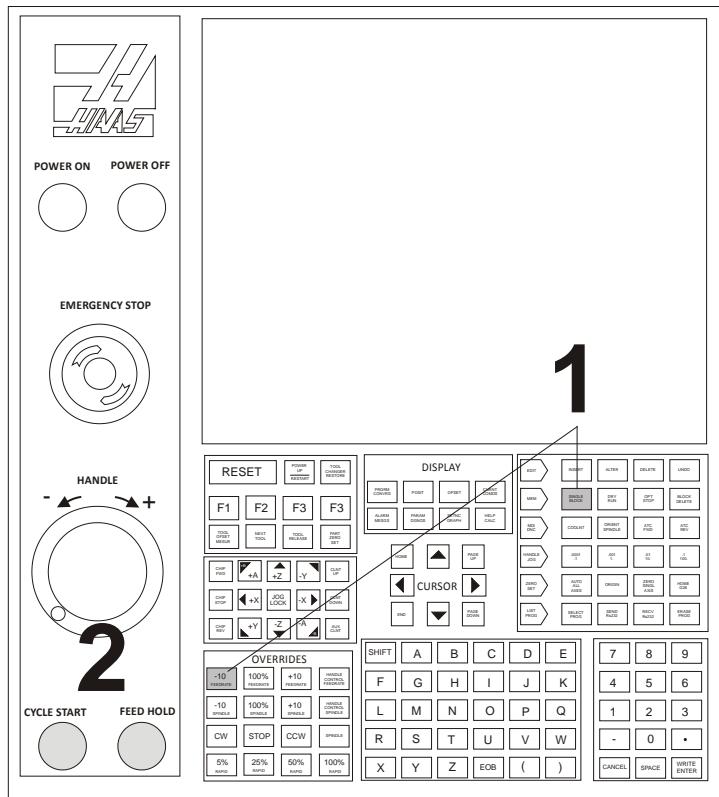
Load CNC Program

Sheet 2 of 2

Run CNC Program

This is the preferred process for running a new program. Once a program is proven, all feed rates can be set to 100% and single block mode can be set to off.

	Check List	Buttons
1	<p>Pre-Start</p> <ul style="list-style-type: none"> ▪ Ensure vise or fixture is secure and that you have a safe setup. ▪ There should be no possibility that the work holding will fail to perform as required. ▪ Remove vise handles. ▪ Clear the work area of any tools or other objects. ▪ Close the machine doors. ▪ Turn Single Block Mode On. ▪ Press Rapid Feedrate -10 button eight times to set rapid Feed Rate Override to 20% of maximum. 	<div style="text-align: center;">  <div style="margin-top: 100px;"> OVERRIDES <div style="display: flex; justify-content: space-around; align-items: center;"> -10 FEEDRATE 100% FEEDRATE +10 FEEDRATE HANDLE CONTROL FEEDRATE </div> </div> </div>
2	<p>Start</p> <ul style="list-style-type: none"> ▪ Place one hand on Feed Hold button and be ready to press it in case there are any problems. ▪ Press Cycle Start Button. 	<div style="text-align: center;"> CYCLE START FEED HOLD  </div>



Caution: A common error is setting the Fixture or Tool Length offset incorrectly. When running a program for the first time, set the machine to Single block mode. Reduce rapid feed rate to 25%, and proceed with caution. Once the tool is cutting, turn off single block mode and let the program run. Do not leave the machine unattended, and keep one hand on the feed hold button. Listen, watch chip formation, and be ready to adjust cutting feed rates to suite cutting conditions.

Adjusting CDC Offsets

Machining operations that use Cutter Diameter Compensation (CDC: G41/G42) can be adjusted to account for tool wear and deflection. Measure across a finished feature on part and compare it with the desired value. Subtract the Actual from the Target sizes and enter the difference into the CDC register on the control for that tool. For example:

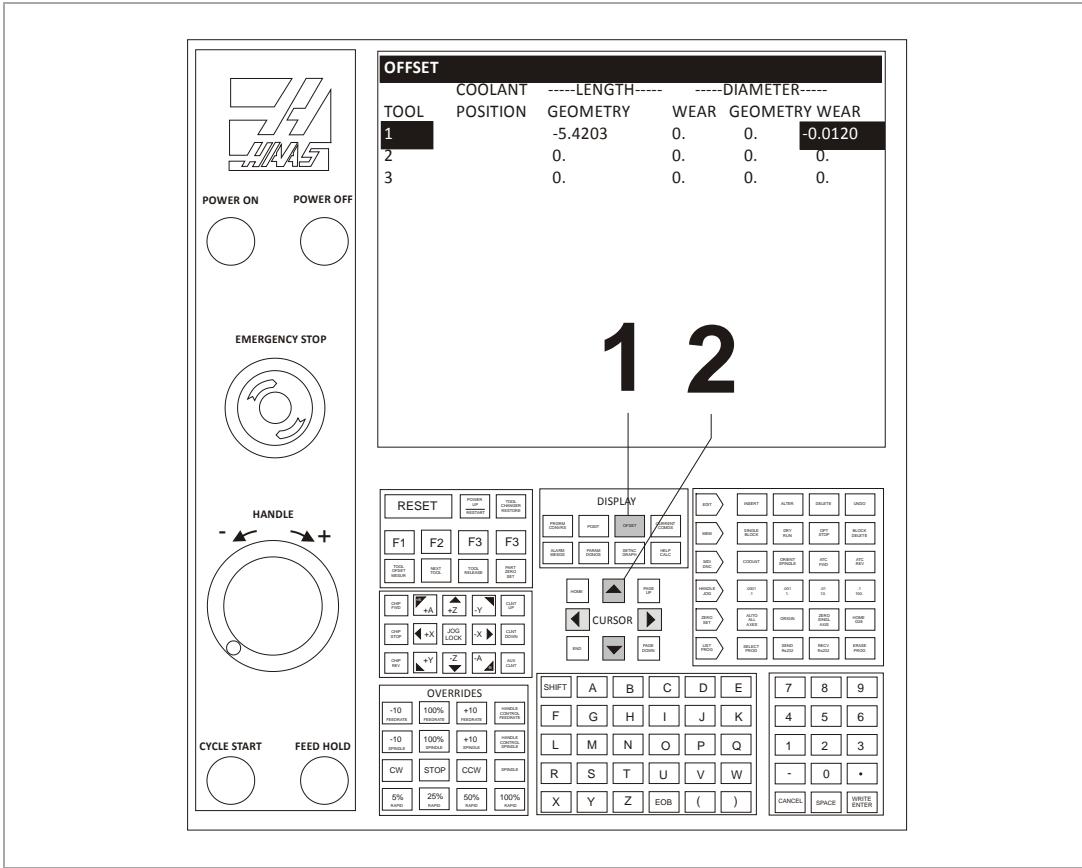
Target Feature Size: 1.0000
Actual Feature Size: 1.0120
Wear Value: -0.0120*

The tool path will now be compensated for the size difference. Running the same operation again should result in the feature being exactly the target size.

*Wear compensation is used only on contour passes. It is not used for face milling, 3D milling, or drill cycles. Select the Wear Compensation option in your CAD/CAM software and, if needed, set a **Tool Diameter Wear** value as shown above. When used, the wear value is always a negative number.*

*Always set **Tool Diameter Geometry** to zero for all tools since CAD/CAM software already accounts for the tool diameter by programming the tool center line path.*

Check List		Buttons
1	Offset Page: Select	
2	Adjust Diameter Offset <ul style="list-style-type: none">▪ Pg Up/Dn to highlight the tool to be adjusted.▪ Enter a value using the numeric keypad.▪ Select	



Adjusting Offsets

Sheet 2 of 2

Shut Down CNC

Check List		Buttons
1	Remove tool from spindle: <ul style="list-style-type: none">▪ Enter the number of an empty tool carousel.▪ Select ATC FWD	
2	Jog Machine to Safe Area: <ul style="list-style-type: none">▪ Select Jog	
3	Shut Down Button: Press	 POWER OFF

Post Power-Down Checklist:

- Wipe spindle with a soft clean rag to remove coolant and prevent rusting.
- Put away tools.
- Clean up work area.
- Always leave the machine, tools, and equipment in the same or better condition than when you found them.

Shut Down CNC

Sheet 1 of 1

Note: It is important to clean the machine after each use to prevent corrosion, promote a safe work environment, and as a professional courtesy to others. Allow at least 15-30 minutes at the end of each day for cleaning. At the very least, put away all unused tools and tooling, wash down the machine with coolant, remove standing coolant from the table, and run the chip conveyor.

Lesson 7

2D Milling Toolpaths

Upon successful completion of this lesson, you will be able to:

- Explain the difference between 2-1/2D, 3D, and 4-axis machined parts.
- Explain the difference between common CAD and CAM graphics views
- Identify 2D machining features based on part geometry and your knowledge of tools and 2D toolpaths.
- Identify commonly used machining parameters for 2D tool path operations.
- Explain the purpose, general parameters, and use of facing toolpaths.
- Explain the purpose, general parameters, and use of 2D contour toolpaths.
- Explain the purpose and rules for using Cutter Diameter Compensation (CDC).
- Explain the purpose, general parameters, and use of pocketing toolpaths.
- Explain the rules for slot milling toolpaths.
- Explain the rules for chamfer milling toolpaths.
- Explain the rules for radius milling toolpaths.
- Explain the rules for drilling and peck drilling toolpaths.
- Explain the rules for tapping toolpaths.

Overview

CNC milling toolpaths are broadly classified as either 2D, 3D, 4-axis, and 5-axis, depending on the number of axes involved and how they move. The term, 2D, is a bit of a misnomer because all modern CNC machines control at least three axis and all three axes move at one time or another for every 2D machining operation. A more accurate term, 2-1/2D, is commonly used in CNC manufacturing.

7.1 - 2D/3D/4X/5X Defined

2D (Prismatic) Parts

2-1/2D milling toolpaths machine only in the XY plane. The Z axis is used only to position the tool at depth. The move to the cutting plane is a straight down feed, rapid, ramp or helical feed move.

The term, Prismatic, is a term commonly used in engineering used to describe 2-1/2D parts. There are, however, prismatic parts that require 4th or 5-Axis machining, so the term is used in machining only to describe parts where all machined faces lie normal to the machine tool spindle. Because 2-1/2D is a clumsy term this book uses Prismatic and 2D interchangeably to describe parts just described, on a CNC mill with three controllable axes (XYZ). The XY axes are normal to the machine spindle and Z is used only to position the tool to depth (either in a feed or rapid motion).

Figure 1 shows a prismatic part. All machined features lie parallel to the XY plane. Each Z-level can be machined by positioning the tool at a fixed Z-level and then moving the XY axes to remove material. Every feature can be reached with the tool approaching either from the Front or Bottom views. There are several cutting planes in this example, including the model top (1), top of the face where the holes start (2), the bottom of the pocket (3) where the slots begin, the bottom of the slots (4), and the bottom of the hole through the center (5).

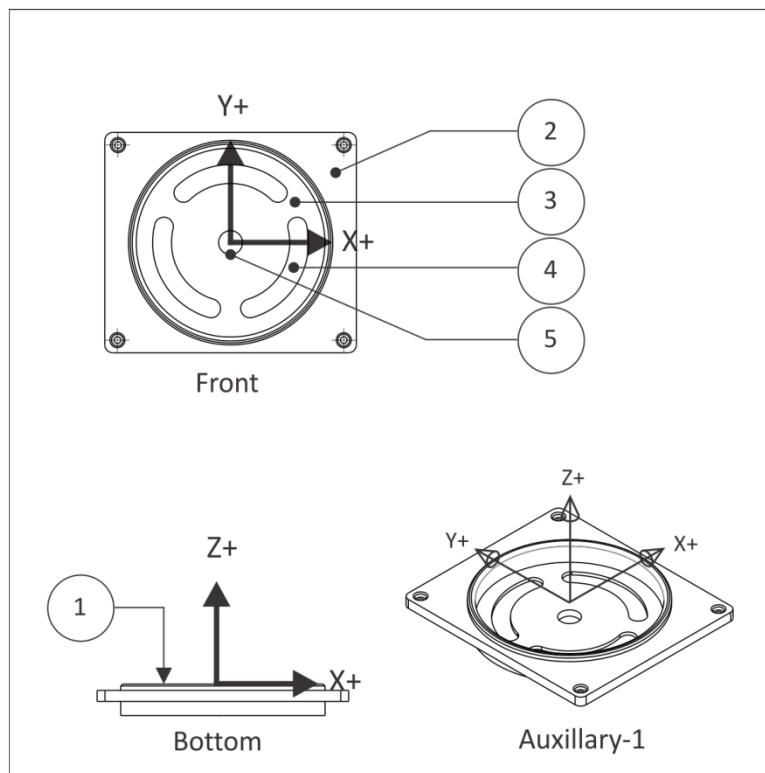


Figure 1: Prismatic Part (Orientation in CAD)

3D Parts

3D refers to non-prismatic parts, including molds and complex organic shapes. Most consumer goods, for example, include 3D features. Figure 3 shows half of a stamping die. This part is typical in that it includes both 3D and 2D features. The 2D features are the top face (1), and the outside contour (2).

3D features, like the revolved surfaces (3) and fillet (4), require more complex machine motion. The revolved surfaces require XZ tool motion. The fillet requires XYZ tool motion. Even the flat (5) and cavity roughing (though technically planar) require 3D toolpaths because the adjacent revolved surfaces and fillet must be considered to prevent gouging the part. The calculations required to calculate these toolpaths are highly complex and the subject of the next lesson, 3D Toolpaths (Chapter 9).

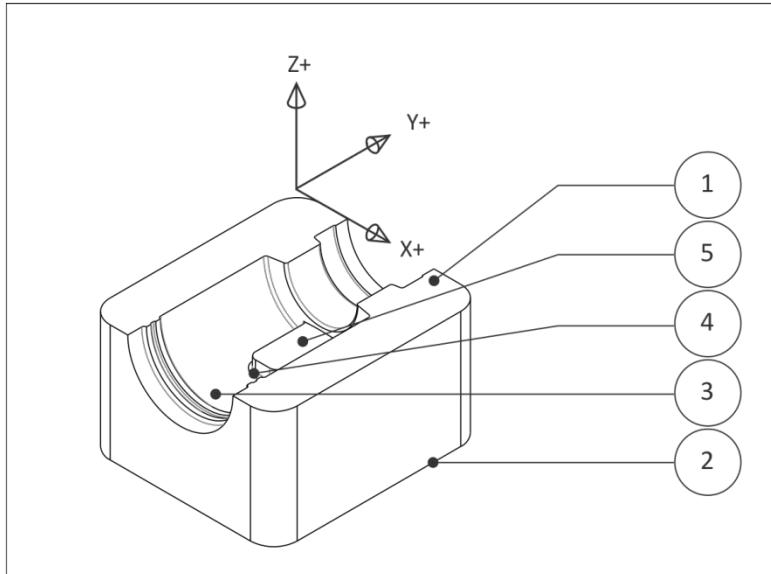


Figure 3: 3D Part

4-Axis Parts

4th axis toolpaths require an auxiliary rotary axis 4th installed on the CNC machine parallel to either the X or Y-axis. 4th axis toolpaths fall into two categories: 4th Axis Substitution and Simultaneous 4th Axis.

Axis substitution paths are illustrated in Figure 4. The most common setup is with the rotary axis mounted parallel to the CNC X-axis. With axis substitution machining, the tool axis centerline always points towards the centerline of the rotational axis (no Y-motion) when milling. The tool feeds to depth (Z) and then only up to two axes can move at once (X-A) to make the feature.

The name ‘substitution’ comes from how these paths were defined before CAD was widely used. Geometry was drawn flat (XY) and then the Y-axis values were converted to A-rotational values, based on the cylinder radius. In other words, the flat geometry was “wrapped” around a constant diameter cylinder; similar to how the SolidWorks® Wrap function works.

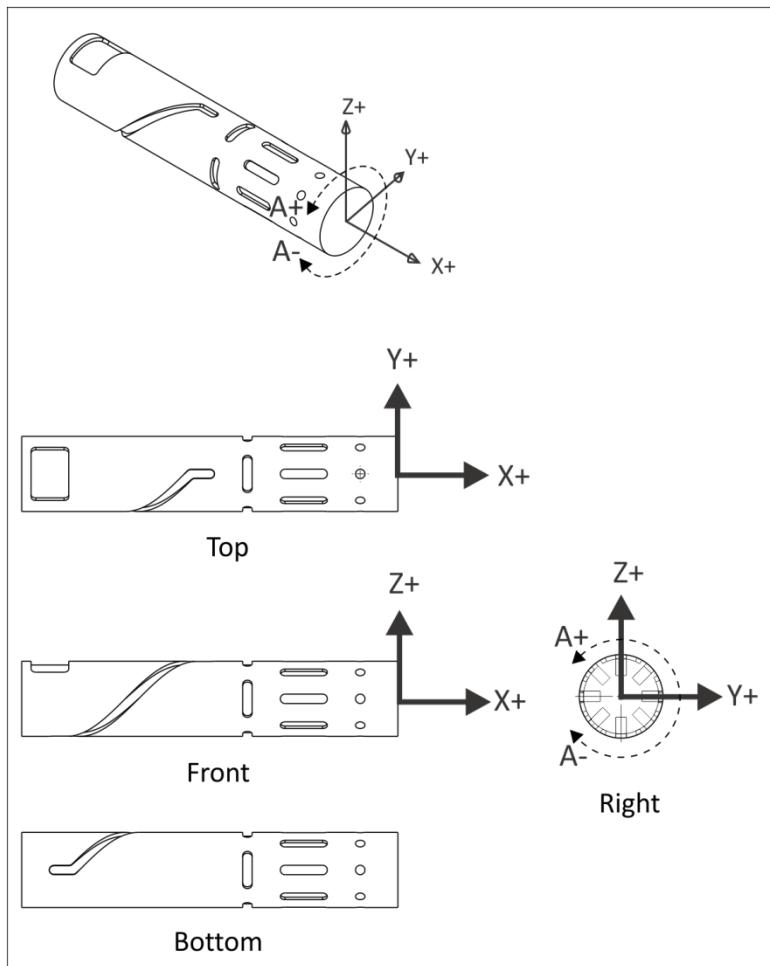


Figure 4: 4th Axis Substitution (X-A)

Simultaneous 4th axis machining allows all 4 axes to move at once (XYZA). This type of motion is very complex and is actually a sub-category of Simultaneous 5-axis machining. 5th Axis parts require all 5 machine axes to move at once (XYZAB).

Simultaneous 4th and 5th axis machining is beyond the scope of this course.

7.2 - Standard CAD Views vs. CAM Views

Figure 1 shows the part oriented as it is modeled in the mechanical CAD software. For CAM, it is helpful to display the part in the same orientation as viewed while sitting on the CNC machine. For a Vertical Machining Center (VMC) this requires updating standard views to look like those shown in Figure 2. The G-code file is generated in relation to the Work Coordinate System (WCS), so changing views is not required for CNC programming. It simply helps visualize CNC machining processes when using CAM.

In SolidWorks to switch views from CAD space (Figure 1) to CAM space (Figure 2) to the following:

1. Open the **View Orientation** dialog and select **Front** (view) so the part displays as shown in Figure 1.
2. Highlight the option **Top**.
3. Select **Update Standard Views**.

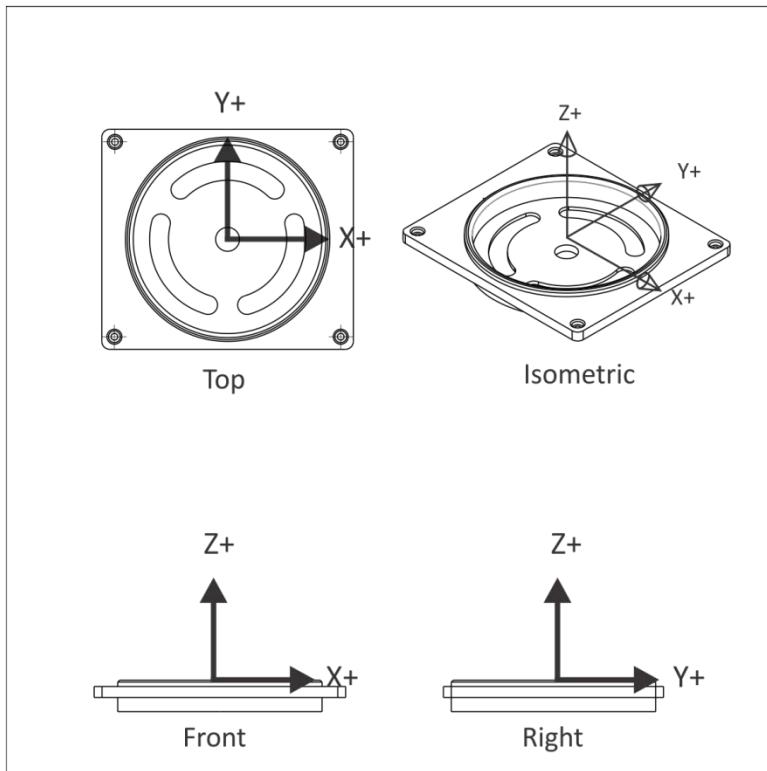


Figure 2: Prismatic Part (Orientation in CAM)

7.3 – CAD Features vs. Machining Features

Parts designed in SolidWorks® are composed of features, including Extruded Cuts, Fillets, Chamfers, and Holes. A CNC milling machine creates these features using machining operations like Face, 2D Contour, 2D Pocket, and various Drilling operations.

Knowing which machining operation to use to make which feature is sometimes obvious. For example, the slots in Figure 3 are created using a Slot Mill pocketing operation, the large extruded cut using 2D Pocket, and the Chamfer using Chamfer milling.

However, sometimes these decisions are not so obvious. For example, the hole through the part center could be created using Drill, 2D Contour, 2D Pocket or Circular Pocket milling. You may wonder, is the large flat (where the holes begin) a 2D Contour or 2D Pocket? Furthermore, which features on this part should be machined from the Top and which from the bottom?

The operations the CNC programmer chooses and their sequence depends on a bewildering number of factors, including feature size, tool used, capabilities of the machine, feature tolerance and how the part is gripped. The rest of this chapter will introduce how to begin looking at 2D parts and begin making CNC process decisions.

To begin with, in most cases you want to first machine the side of a 2D part that has the most features; finishing as much of the part as possible with the first CNC setup. This is often the Front view of a part designed in SolidWorks. In this example, that means machining the side with the slots first (Front CAD view) rather than the opposite side.

7.4 Toolpaths By Type and Use

Before going further, it is helpful to understand how 2D toolpaths are classified in most CAM software. Table 1 lists the common 2D toolpaths by type and common use. For example, 2D contour, chamfer, and fillet toolpaths are often accomplished using the 2D Contour menu selection. Of course, where each function is located will be slightly different depending on the CAM product, but this list is appropriate to most modern CAM.

Type	Toolpath	Common Uses
Face	Face	<ul style="list-style-type: none"> Finish face of part.
	Island Facing	<ul style="list-style-type: none"> Finish face with open sides and bosses.
2D Contour	Contour	<ul style="list-style-type: none"> Loops. Partial loops. Single edges. Stick (single point) fonts. Create dovetail, keyset, or saw cut.
	Chamfer	<ul style="list-style-type: none"> Create chamfer using tapered mill or center drill. De-burring.
	Fillet	<ul style="list-style-type: none"> Creating fillet using Corner Round tool.
Pocket	Pocket	<ul style="list-style-type: none"> Remove excess material. Machining TrueType (outlined) fonts and logos.
	Slot Mill	<ul style="list-style-type: none"> Straight slot. Arc slot.
Drill	Drill	<ul style="list-style-type: none"> Create spot drill, drill, tap, bore or reamed hole.
	Circular Pocket Milling	<ul style="list-style-type: none"> Making holes greater than .75in diameter.
	Thread Mill	<ul style="list-style-type: none"> Create ID threads over .75in diameter. Create milled OD threads of any size.

Table 1: 2D Toolpaths and Common Uses

It is probably obvious to you now that manufacturing is an exceedingly complex process. Many factors influence every decision and often more than one solution to any problem. Manufacturing is also a win/lose game. Either the part is right (within tolerance) or not. Don't be overwhelmed by the choices for now. Some knowledge and experience will help you settle many of these variables and greatly simplify the job of planning CNC processes.

7.5 - 2D Machining Features Example

Figure 5 and Table 2 shows the operations and machining sequence to CNC mill this part.

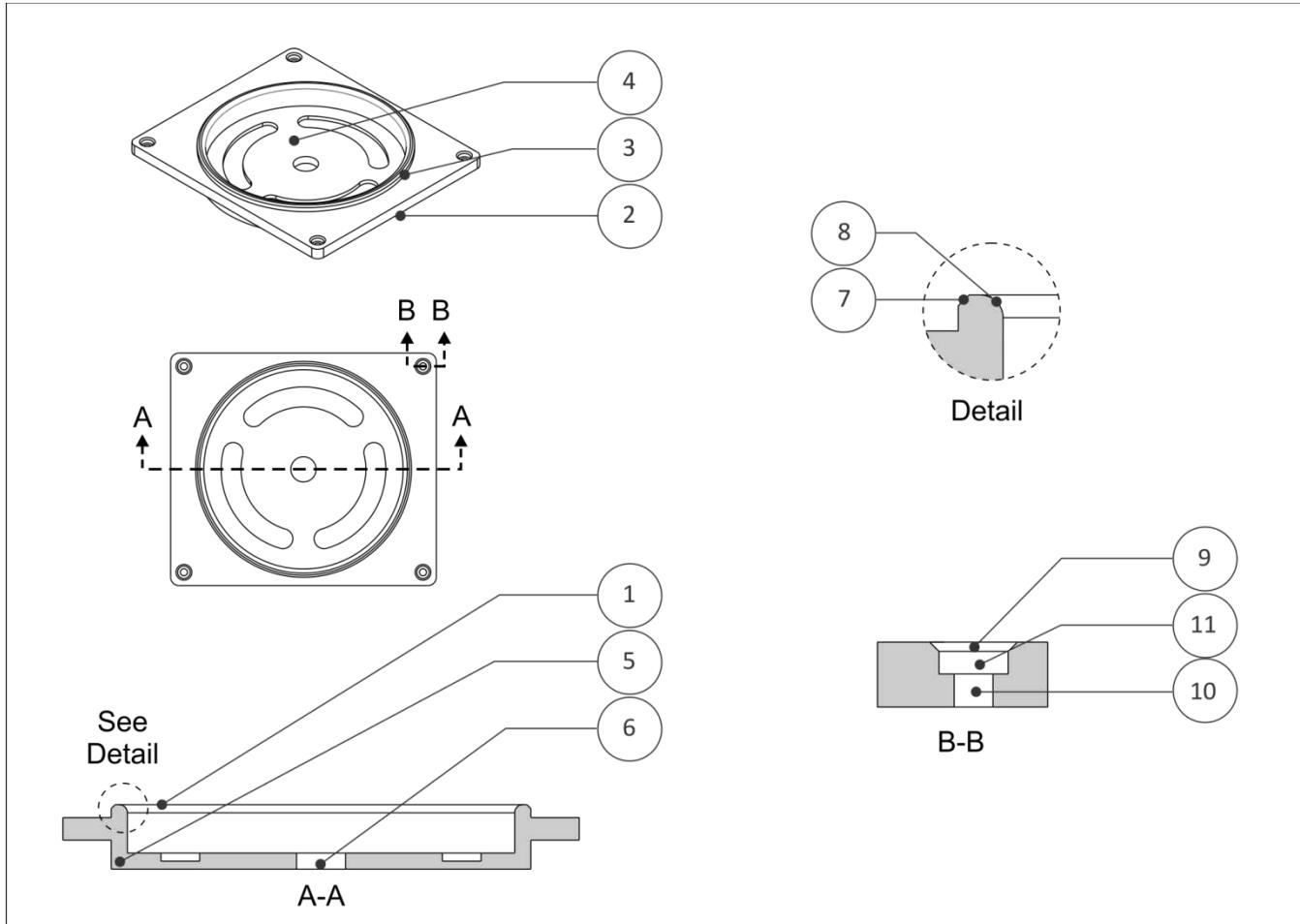


Figure 3: 2D Machining Features Example

Op.	Toolpath	Notes
1	Face	It is common practice that the first machining operation roughs and finishes to the highest flat surface of the part. Face paths overlap the sides of the loop selected.
2	2D Contour	Machine outside loop.
3	2D Contour	Machine outside of boss.
4	2D Pocket	Use Pocket to rough and finish enclosed loops.
5	Slot Mill	Mill slots.
6	Circular Pocket Mill	Machine the center hole through. You could also use a Drill operation to make this hole, but would center-drill the hole first.
7	Chamfer	Use 2D Contour, Chamfer function and a chamfer tool (or center drill) to make this 45 degree

		chamfer.
8	Fillet	Use 2D Contour, fillet function and a corner round tool to make this fillet feature.
9	Spot Drill	Spot drill all holes to: 1. Ensure subsequent drill does not wobble and thus is located precisely. 2. Create chamfer for this hole.
10	Drill	Drill to make hole. Do this before the Circular Pocket Mill so the Spot Drill conic still exists.
11	Circular Pocket Mill	Create counterbore.

Table 2: 2D Features Example

7.6 - 2D Toolpath Terminology

Though the terminology and ways of working vary widely, all CAD/CAM software needs the same basic information to function. Figure 4 shows parameters common to 2D tool paths.

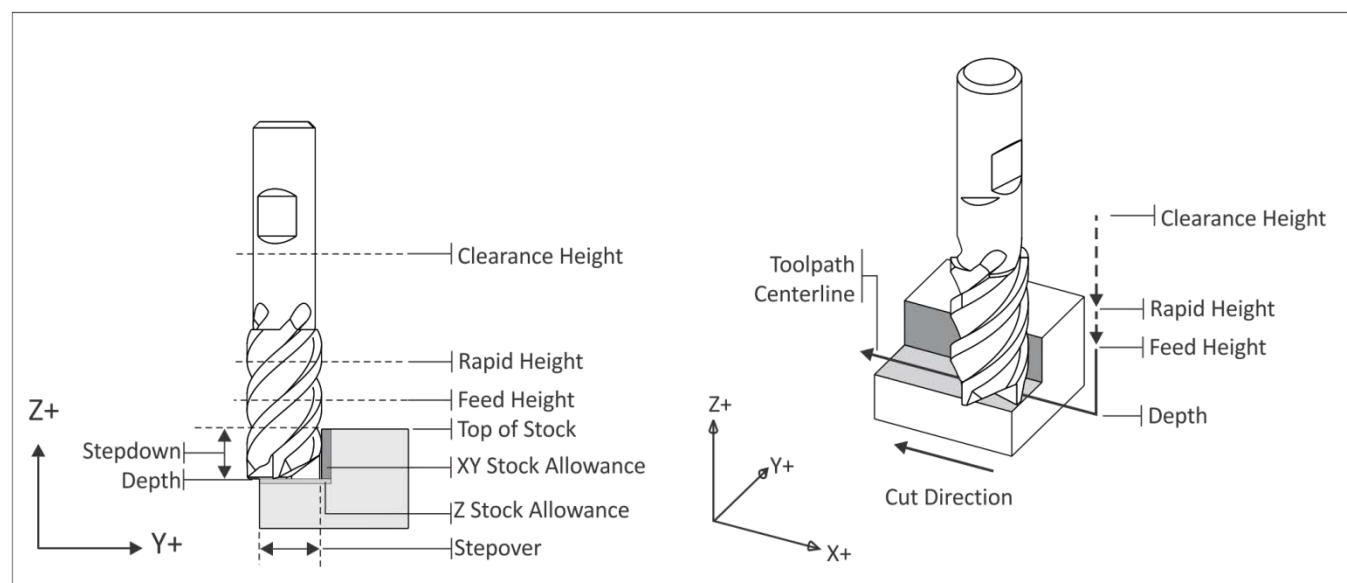


Figure 4: 2D Tool Path Terminology

Clearance Height is the first height the tool rapids to on its way to the start of the tool path. It is usually set 1.000in above the top of stock because this makes it easier to see if the tool length offset register was set properly.

Rapid Height is the second height the tool rapids to, and the height the tool retracts to between moves (unless set higher to clear clamps). It is usually set to .250in above the top of the finished part face.

Feed Height is the last height the tool rapids to before starting to feed into the cut. It is usually set to .1000in above top of stock. No rapid motion occurs below this height.

Top of Stock is the top of the finished face of the part. This value is used as the reference plane for depths.

Stepdown is the depth of material removed with each cutting pass. This illustration shows one pass, but for deeper cuts or harder materials, many passes may be required to cut to the final depth.

Depth is the final cutting depth of the machining operation.

Stepover sets how much material the tool removes with each pass in the XY direction.

XY Stock Allowance is the material remaining on the finished wall of the part to be removed by subsequent operations.

Z Stock Allowance is the material remaining on the finished floor of the part to be removed by subsequent operations.

Toolpath Centerline represents the actual coordinates in the CNC program. In this book, rapid moves are shown as dashed lines and feed moves as solid.

Facing

Facing is often the first machining operation. It is used to cut away excess material and finish the highest flat face of the part. Depending on how much stock is removed, several roughing cuts may be required. A smaller finish pass ensures a flat surface and good surface finish.

Use a face mill when possible for all but the smallest part. The large diameter of facing mills and multiple carbide insert cutting edges provide for very high material removal rates.

High speed loop transitions between cut passes produce a fluid tool motion that place less stress and wear on the CNC machine.

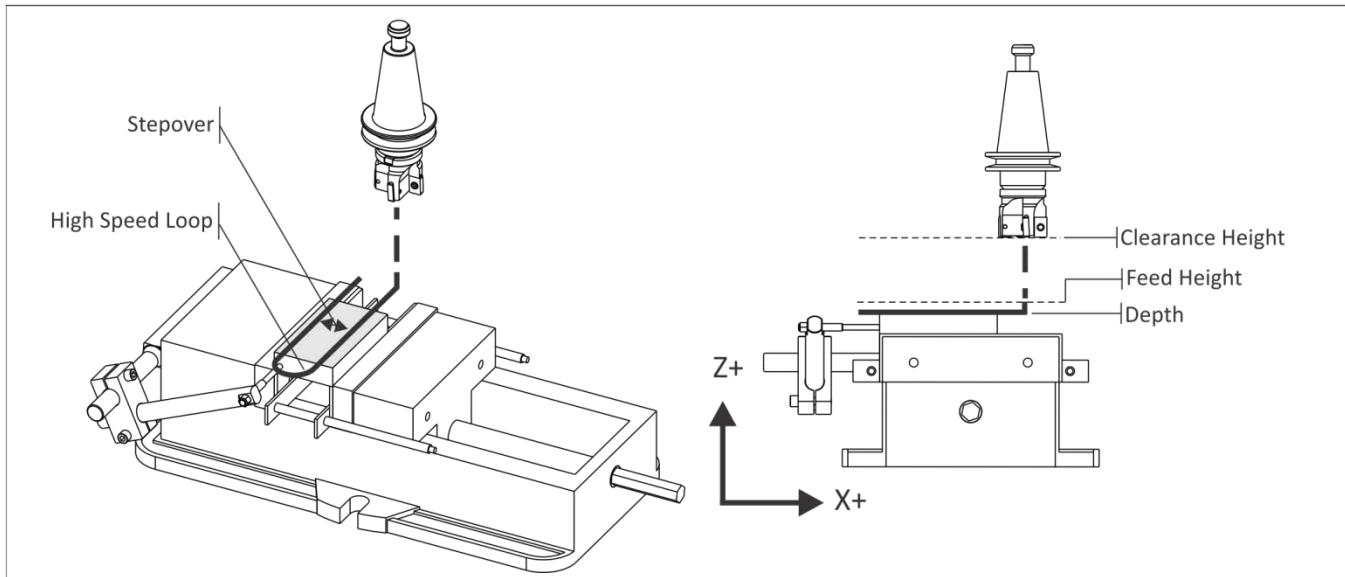


Figure 5: 2D Facing Toolpath

Rules for Facing:

- Because face mills do not plunge well, start the tool path far enough away from the part so the tool does not plunge into the stock material.
- Be aware that saw cut stock can vary considerably in thickness from one part to another: as much as .05/in or more. When planning roughing passes, be sure to account for the worst case stock material – maximum height and add additional roughing passes as needed. It is better to have a “air cut” or two with the shortest stock than to have the tool engage too much material for the highest which could cause the tool to break or the part to be pushed out of the vise or fixture.
- Facing tool paths do not use cutter compensation (CDC).

2D Contour

Contour operations are used to rough and finish outside part walls as shown in Figure 6. Use Cutter Diameter Compensation (CDC) on high tolerance features so the tool path can be adjusted at the machine if needed to account for tool wear and deflection.

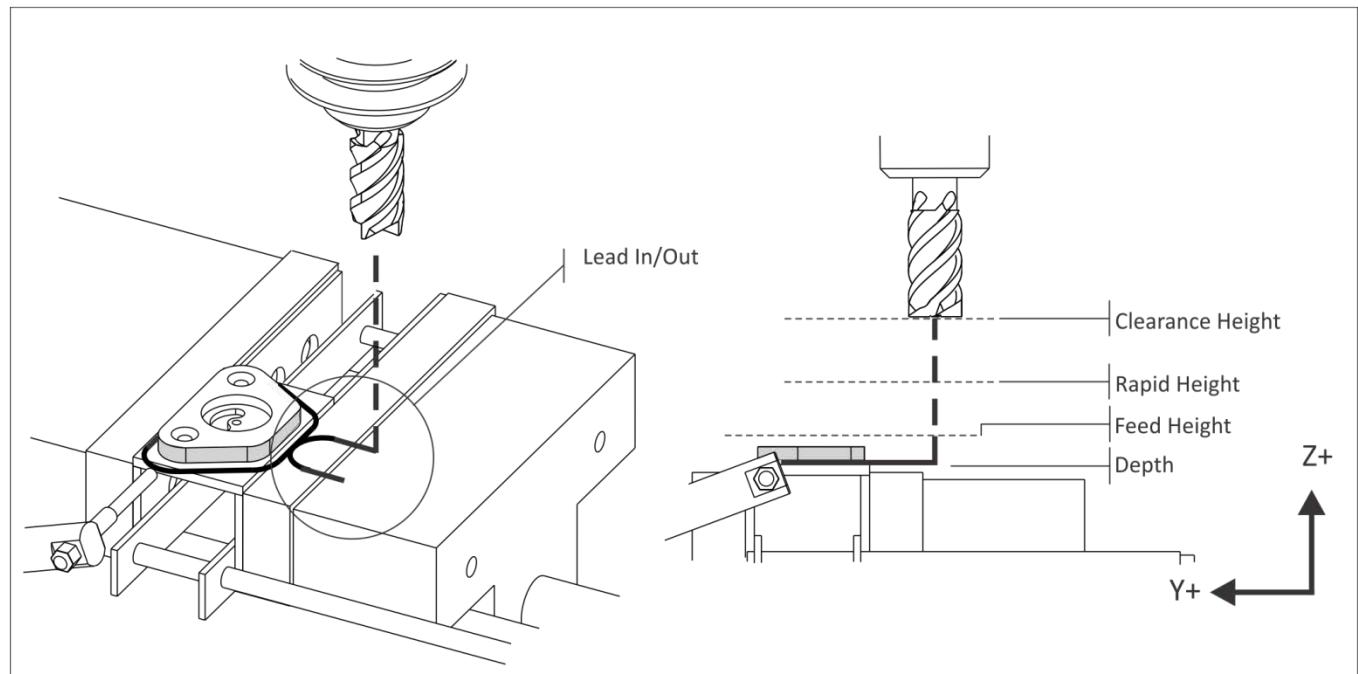


Figure 6: 2D Contour Tool Path

Rules for Contouring:

- Only use CDC when needed. If using new tools and conservative machining parameters, features will likely be within .005 inches of the programmed path without adjustment.
- Start the tool path off the part to allow CDC to be fully in effect for the entire operation. The combined line-arc lead-in/out moves shown in Figure 4 work for most contours. The line is for activating/deactivating compensation, and the arcs blend the path into the part wall smoothly.
- Set a rapid height value to clear all clamps or other obstacles between cuts.
- Rough the walls and leave a constant thickness of material for the finish operation. This ensures even cutting pressure on the finish pass and thus a more accurate part.
- Extend the cut depth of full walls slightly below the bottom of the wall, but be careful not to cut into the machine table or vise hard jaws! This way, when the part is flipped over to face the other side, no flashing will be left on the bottom of the walls.
- Mill tools cut well in the XY direction, but not as well when plunging in Z. When possible, plunge the tool away from the part to avoid Z-moves into the stock material.
- When taking multiple depths of cut, make the last pass at full depth to remove any marks left by previous depth cuts.
- For tall walls, consider taking one additional finish pass. This so-called “spring pass” follows the same path twice to ensure the walls are perfectly straight and not slightly tapered due to cutting pressure which causes the tool to bend.

Cutter Diameter Compensation

Cutter Diameter Compensation provides a way for tool paths to be adjusted at the machine to compensate for tool wear and deflection. Figure 5 shows how CDC Right (G41) causes the tool to veer to the right of the programmed path.

The compensation value is found by measuring the part feature and subtracting the actual dimension from the desired dimension. The difference is entered in the control CDC register for the tool. The next time program is run, the tool will be offset by this value.

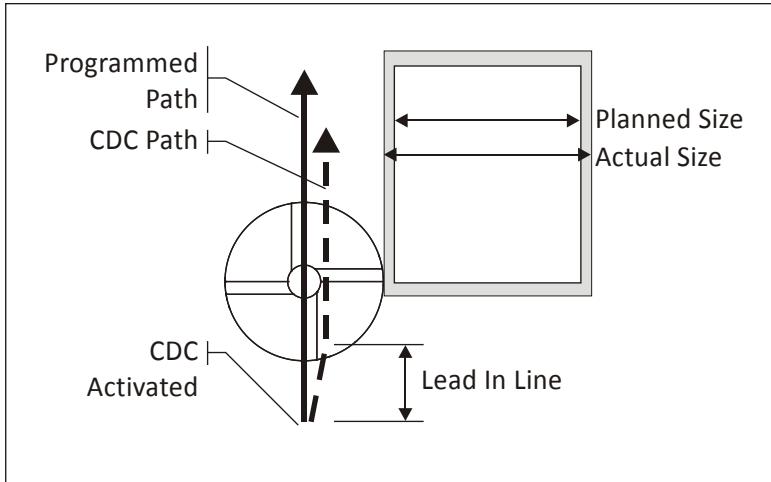


Figure 7: Cutter Diameter

CDC must be turned on or off with a line move, never an arc. Commanding G40/G41/G42 with an arc move will cause a diameter compensation error that will stop the program.

CDC is activated at the end of the line on which it is called, as shown in Figure 5. Notice how the tool moves at an angle from the start to end of the lead-in line. Activate CDC while the tool is away from the part so this angle move happens away from the finished part surfaces. The line-arc moves shown in Figure 4 provide ample clearance for the tool for this purpose.

Pocketing

Pocket tool paths are used to remove excess material. An example of a spiral pocket with helical entry is shown in Figure 6. CDC is not active during the roughing cuts, but may be used for finish passes on walls.

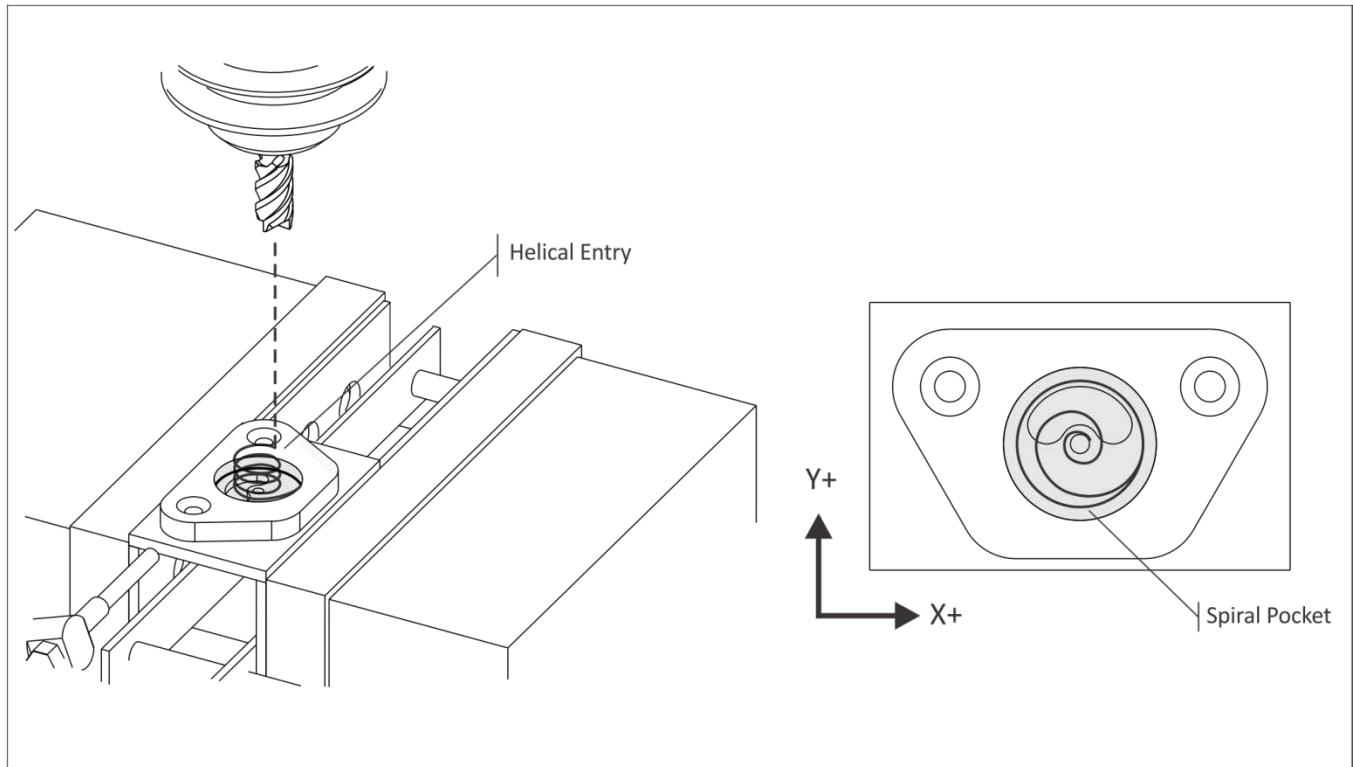


Figure 8: Pocketing

Rules for Pocketing:

- Rough passes should leave a constant thickness of material on the walls and floor of the pocket to be removed by the finish passes.
- Consider using a roughing end mill to remove most of the material. These serrated mills can remove material at a far faster rate than finish end mills. They do leave a poor finish on the floors and walls that must be finished with a separate finish tool and operation.
- Helical moves are a good method for entering a pocket. If space does not allow a helical entry, use a center-cutting end mill or plunge the tool through an existing hole, or a pilot hole created for this purpose. The pilot hole must be at least 50% of the tool diameter.
- Spiral pocketing paths that start near the center of the pocket and move outward in a counter-clockwise direction are best because they cause the tool to continually climb cut.
- Use CDC only on finish passes.

Slot Milling

Slots may be machined using the CAD/CAM contour, pocket, or specialized slot milling functions. In HSMWorks, use 2D pocket and select the slot geometry. HSMWorks recognizes the slot feature and applies a slot milling strategy.

Rules for Slot Milling:

- Use a tool smaller than the width of the slot whenever possible.
- A ramp plunging move as shown in Figure 7 is the most efficient way to mill a slot.

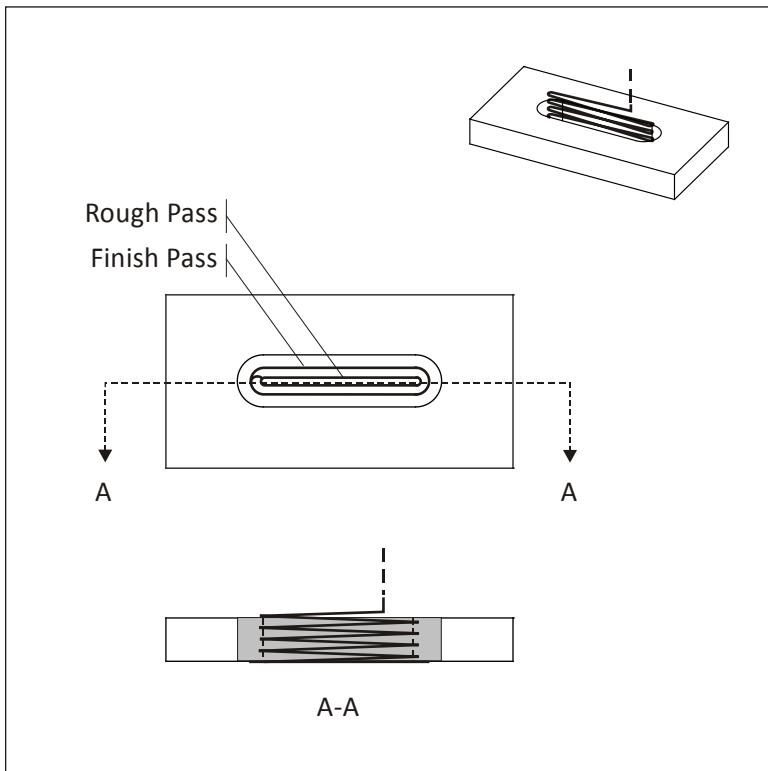


Figure 9: Slot Milling

Chamfer Milling

Chamfer is a type of 2D contour milling. Chamfer mills are of various tip angles are in high speed steel, carbide, or as insert type tools.

Rules for Chamfer Milling:

- Because the tip of a chamfer mill is not a sharp point the width of the chamfer may be wider than expected if you set the tool like an end mill. To prevent cutting too deep, consider raising the TLO about .010 inches after setting it. Then machine the chamfer, check its size, and adjust the TLO down as needed to produce the correct width chamfer.
- Offset the chamfer mill as shown in the magnified view below to move the tool tip is away from the bottom of the chamfer. This ensures a clean bottom edge and, because tool rotational velocity increases with tool diameter, is a more efficient to use the tool.
- Chamfer with a spot drill to precision de-burr sharp corners.

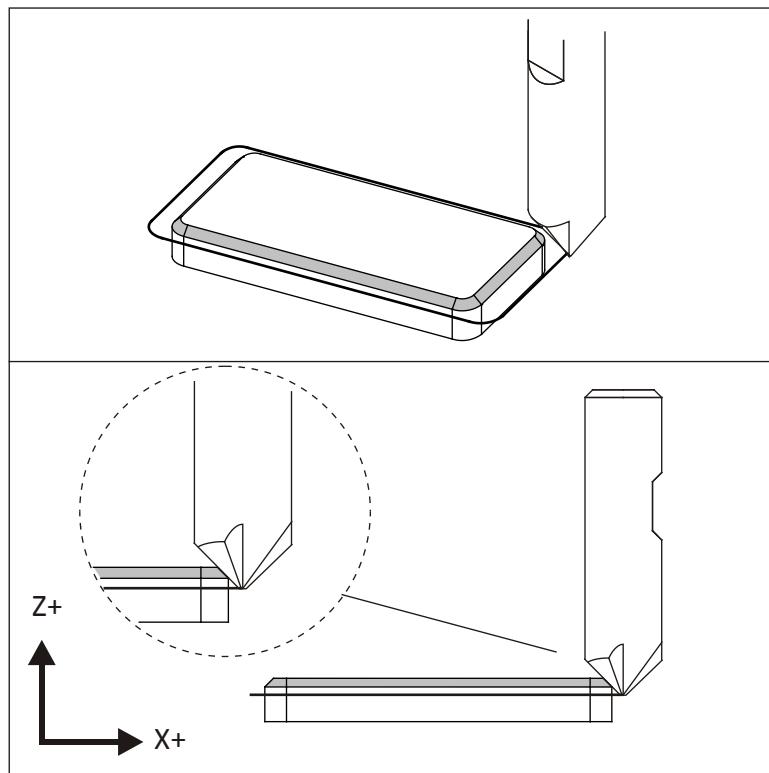


Figure 10: Chamfer Milling

Radius (Corner-Round) Milling

Radius milling is a form of contour milling. Corner round tools are available in high speed steel, carbide, or insert type tools.

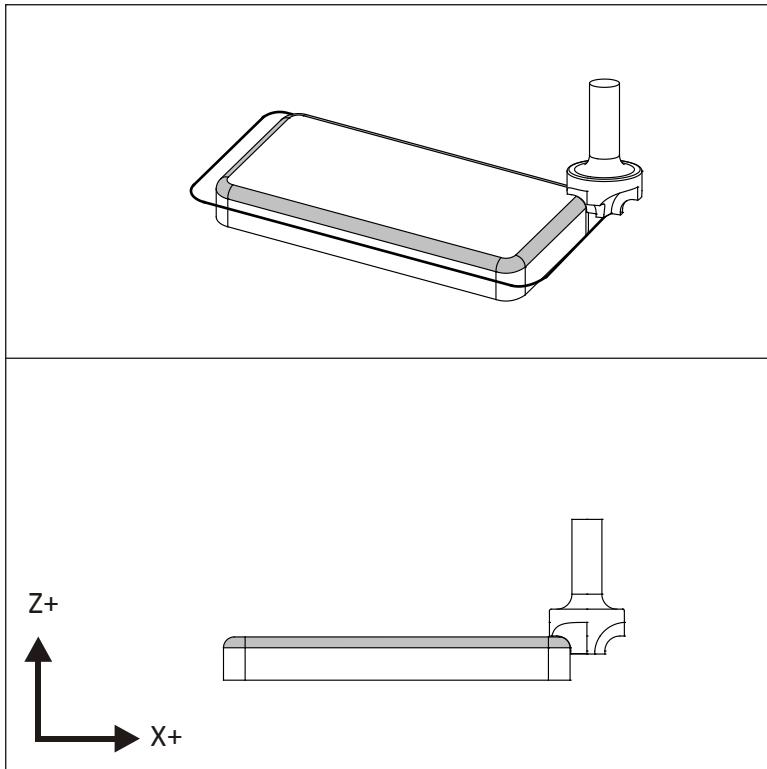


Figure 11: Radius Milling

Rules for Radius Milling:

- The horizontal and vertical cutting edges of a radius mill are sloped slightly to blend the radius into the walls.
- Take two finish passes to improve surface finish.
- Another way to form a corner radius is to use a ball mill and 3D contour tool path. This method saves purchasing a radius mill and is suitably efficient for prototype and small production manufacturing.

Center Drill

Center drills create a conical cut on the face of the part. This helps prevent subsequent drill tools from wobbling and thus ensures they will be positioned precisely.

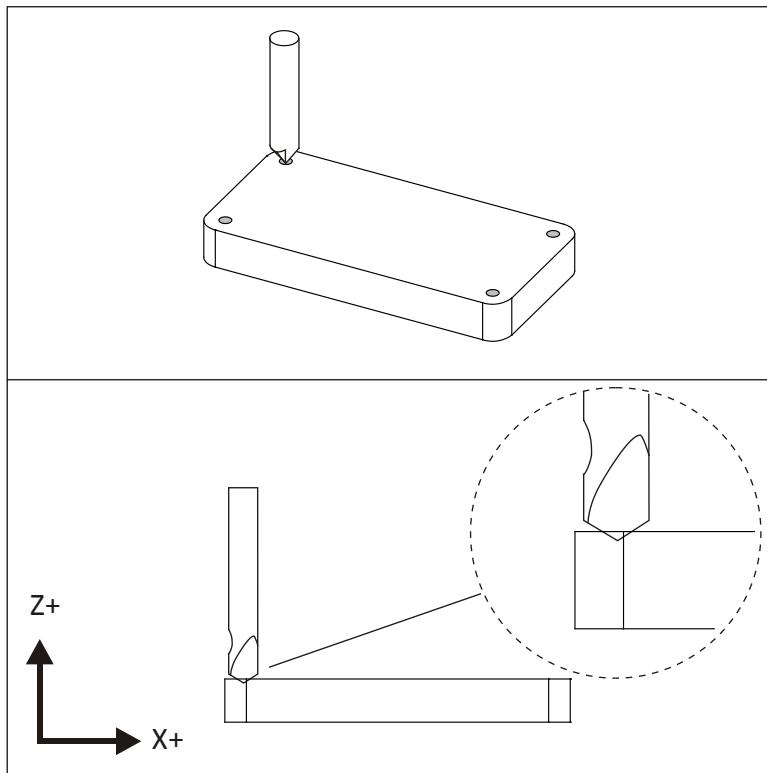


Figure 12: Center Drilling

Rules for Center Drilling:

- A good rule of thumb is to use a tip depth equal to the radius of the subsequent drill hole.
- Use a combination center-spotting drill for spot faced holes.

Drilling

Holes that are less than the diameter of the drill can be created with a single plunge move. Deeper holes use a Peck Drill cycle where the tool is retracted after removing a small amount of material (typically .050 inches).

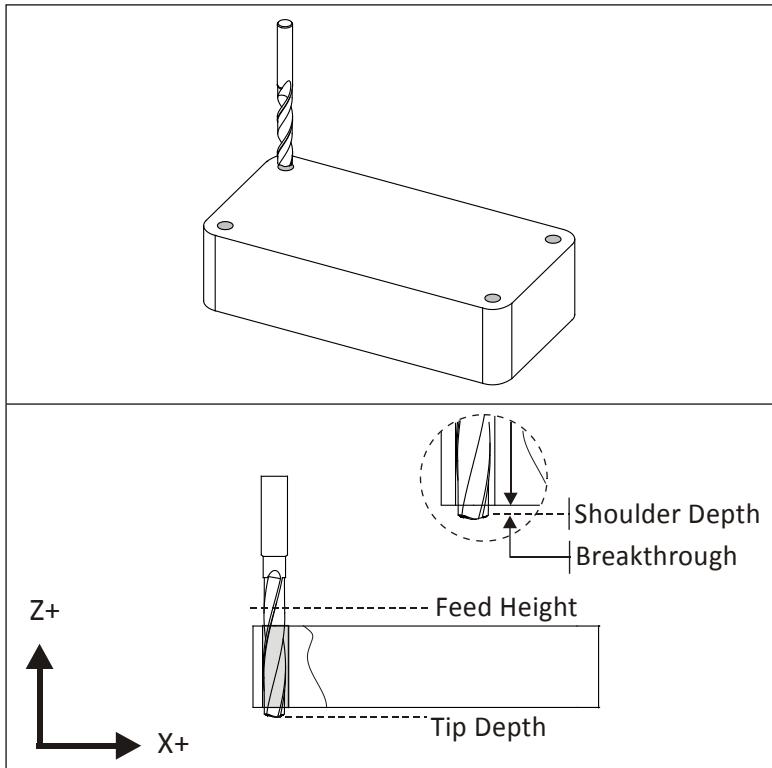


Figure 13: Drilling

Rules for Drilling:

- Center drill all holes to ensure they are located precisely.
- Peck drill (G83) holes that are deeper than the diameter of the drill. Full retract peck drill cycles take more time than partial retracts, but minimize the chance of tool breakage.
- CNC Programs control the tip of the tool. Be sure to provide additional depth to compensate for the tool tip and include a breakthrough allowance to prevent a flange or burr on the bottom.

Tapping

Tap cycles are similar to simple drill cycles except the feed and speed are coordinated to properly match the thread lead. CAD/CAM software calculates the feed according to the cutting speed and threads per inch (TPI) of the tap.

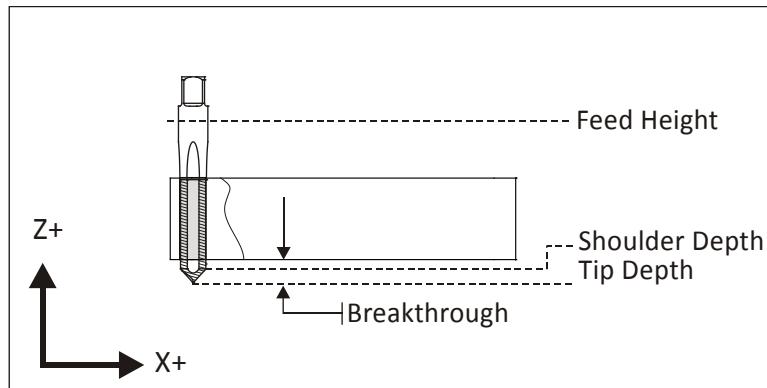


Figure 14: Tapping

Rules for Tapping:

- Use the drill chart in Appendix A to find the correct drill diameter for cutting taps. Use the manufacturers recommended drill size for form taps.
- Consider brushing on tapping fluid instead of using coolant for small tapped holes to help prevent the tap from breaking.
- Tapped holes smaller than about #8 can be difficult to create on the machine without breaking the tap. Consider tapping these holes by hand rather than on the CNC.
- If the machine does not support Rigid Tapping, a floating tap holder is required to tap. Refer to the tapping head documentation for proper use.
- Be sure to specify a tip depth sufficient to account for the tip and initial taper of the tap.
- Older machines may require a larger feed height to allow the machine spindle to reach full speed before the thread engages the material.

Lesson 8

CNC Turning

Upon successful completion of this lesson, you will be able to:

- Identify the elements of the Lathe coordinate system.
- Identify common lathe work holding.
- List the most commonly used Lathe tools.
- Calculate lathe speeds and feeds.
- Describe Constant Surface Speed (CSS) speed control.
- Explain the purpose, general parameters, and use of lathe tool paths, including face, turn, groove, cut off, thread, and drill cycles.
- Explain the purpose and general configuration of live tools.

Overview

Milling machines work by moving a spinning tool over a stationary part. Lathes work by spinning the part and moving the tool, making them ideal for round parts like shafts, pins and pulleys.

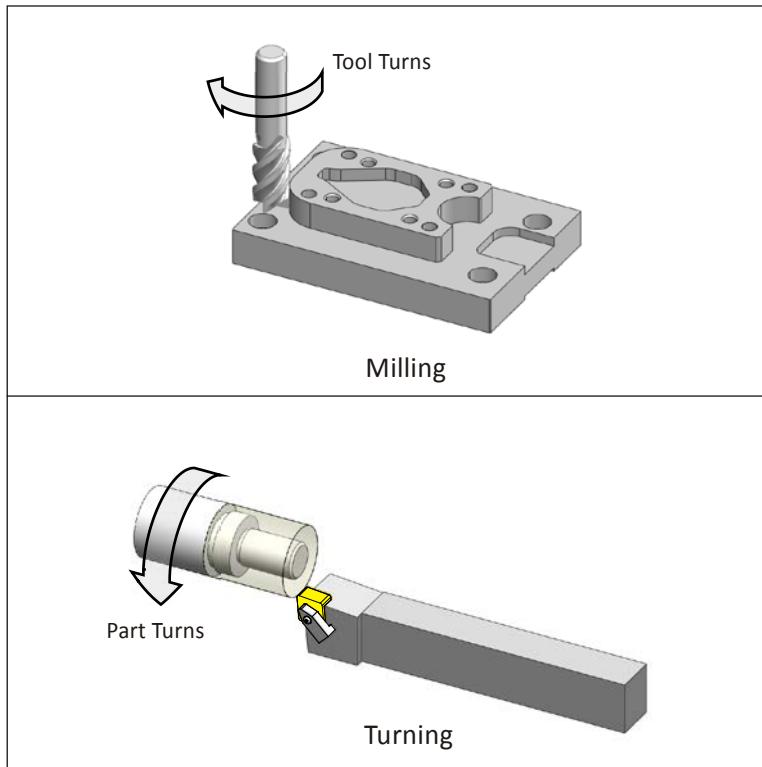


Figure 1: Milling vs. Turning

There are many different configurations of CNC lathes. Some have two spindles, some two tool turrets, and some even integrate milling or grinding in the same machine. There is also variation between similar machines. Horsepower, the size of stock that can be gripped, and how tools are held vary greatly between lathes.

Discussing every lathe configuration is beyond the scope of this book. This chapter discusses one of the most common lathe configurations, called a Turning Center, with one spindle and tool turret.

8.1 - Lathe Components

The major components of a CNC turning center are shown in Figure 2.

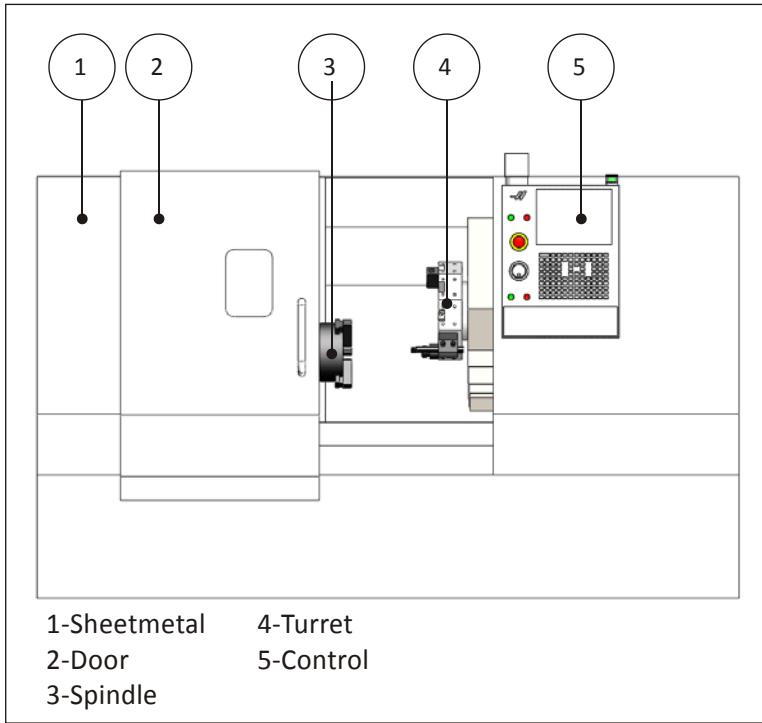


Figure 2: CNC Lathe

1 – Sheetmetal

Protective housing that contain cutting chips and capture coolant for recycling.

2 – Door

The door is closed during operation. Lathes can be dangerous if the part is thrown or a tool breaks during machining. The window is made from a special high impact glass. The lathe should not be operated if this glass is cracked.

3–Spindle

The spindle is attached at one end the machine drive system. The other end attaches the chuck, which grips the part.

4–Turret

The turret holds and moves the tools. Tools are bolted to the turret using a variety of specialized holders, depending on the type of tool. The turret indexes to present the tool to the work piece.

5–Control

The CNC control used to operate the machine.

Spindle

The spindle turns the chuck. The chuck grips the part using hard jaws, soft jaws, or collet. The most common configuration is the three jaw chuck, shown in Figure 3. The chuck requires air pressure to open and close the jaws, and set the gripping force. Pressure must be high enough to securely hold the part, but not so great as to deform fragile parts.

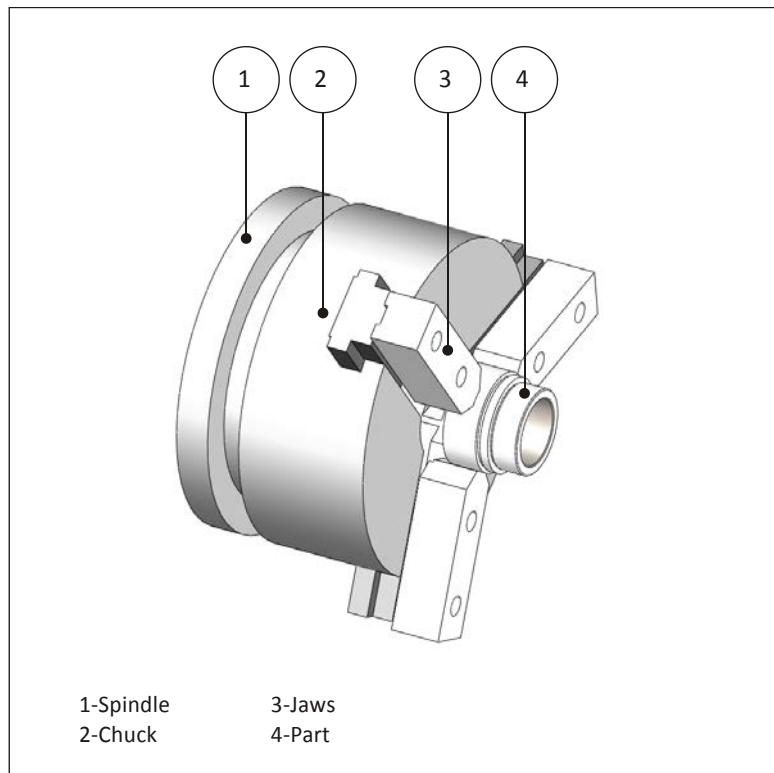


Figure 3: Spindle Detail

Turret

Tool holders bolt to either the front or perimeter of the turret. Tool changes are made by the machine indexing the turret to place the appropriate tool closest to the part.

The method by which the tools are attached to the turret, and the direction the tool faces in relation to the part, vary depending on the tool, operation, and cut direction. For example, a facing tool is oriented radially to the part, to maximize tool rigidity and work envelope. A boring bar is oriented axially to allow the bar to enter and exit the bore.

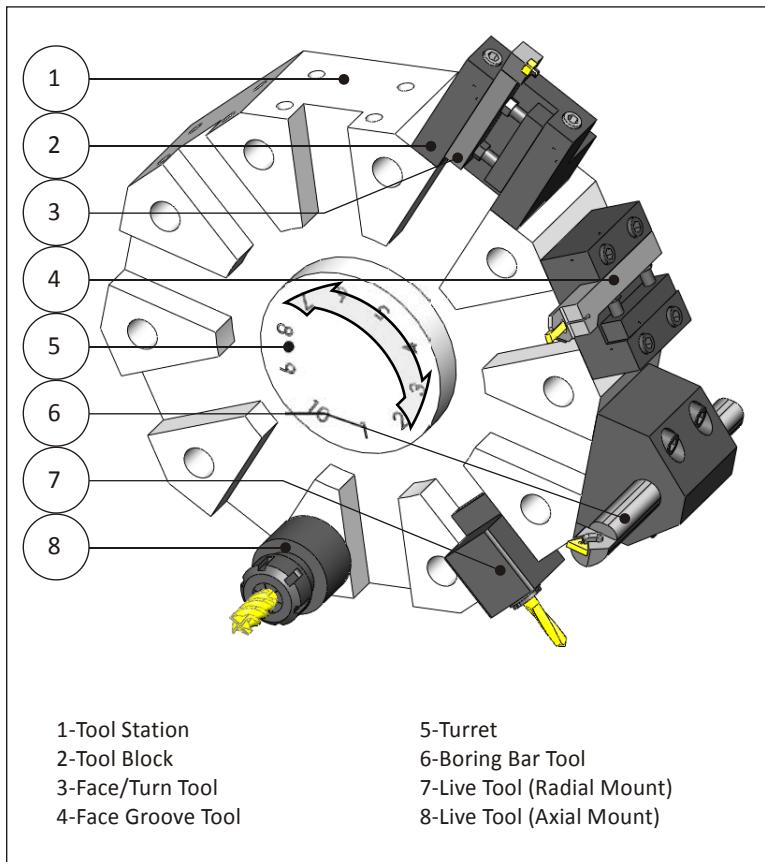


Figure 4: Turret Detail

1 – Tool Station

The turret is divided into stations evenly spaced around the perimeter. Most lathes with tool turrets have about 10 tool stations. Tools are connected to the turret by a tool holder and tool block. The tool holder and block used depend on the type of tool and mount direction.

2 – Tool Block

Tool blocks act as the interface between the tool holder and the turret. They bolt to the face or perimeter of the turret. Different blocks are used depending on the type of tool and orientation.

3-Turn Tool

Turning tools, which includes face, OD rough and finish, groove and cutoff, are usually mounted radially with respect to the part. The cutting tool is usually a ceramic insert mounted in a tool post designed for the specific shape and size insert.

4-Face Groove Tool

Face groove tools are mounted axially from the part.

5-Turret

The turret holds and moves the tools. To change tools, the turret unlocks, rotates to present the active tool to the work piece, and then locks again. Care must be taken that the turret is away from the part so that none of the tools collide with the part as the turret indexes.

6-Boring Bar

A boring bar is used to create a precision size and finish hole through the bore of the part. These are mounted axially with the spindle

7-Live Tool (Radial Mount)

A “live tool” is a tool that rotates, being driven by a mechanism in the holder. Radially mounted live tools are used for cross drilling or milling on the diameter of the part.

8-Live Tool (Axial Mount)

Axial mounted live tools mill or drill on the face of the part.

8.2 – Lathe Coordinate System

Most lathes are 2D machines based on a Z-X Coordinate System. As shown in Figure 5, the Z-axis is parallel to the machine spindle and the X-axis is perpendicular to the spindle. Normal spindle rotation is counter-clockwise, though direction can be reversed for left-handed threads just like with a mill by commanding the proper G-code.

Notice how the turret is set at an incline from horizontal. This so-called “slant bed” configuration keeps cutting chips from accumulating on flat surfaces inside the machine.

Also notice that the tool approaches from the top of the part. This is known as an upper-turret configuration, and requires the tool be placed in the holder upside-down as shown in the inset view in Figure 5.

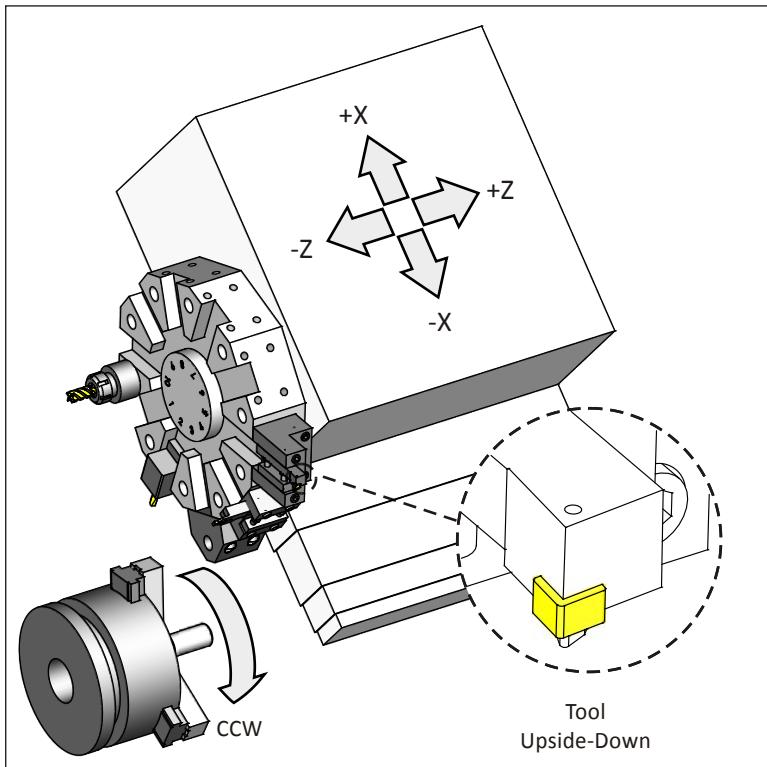


Figure 5: Lathe Coordinate System

8.3 - Driven Tools

One variation of the CNC lathe is called a Mill-Turn center. In the milling mode, the spindle becomes a rotary axis that can stop, index, or move in sync with the other axes to create shapes including face milling, radial holes and slots. For example, Figure 6 shows a face milling operation. The X-C axes are coordinated to move the tool in a square shape on the part face.

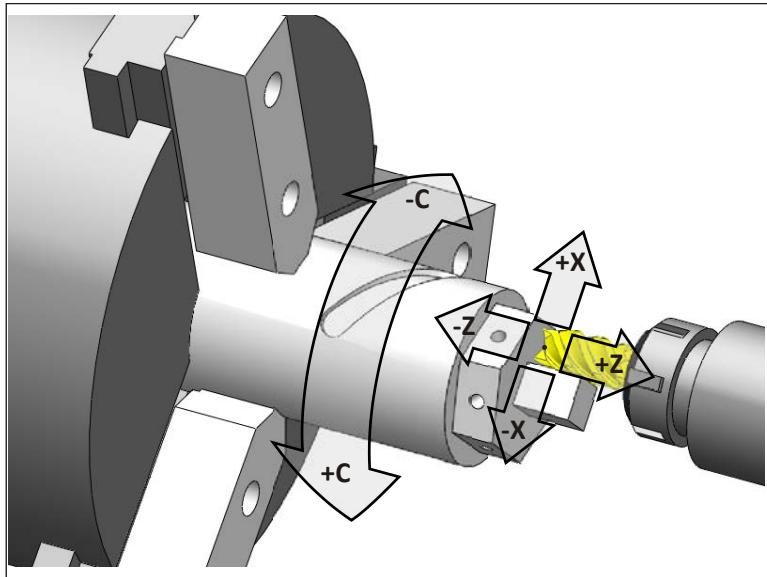


Figure 6: Driven Tool Machining

Mill-Turn centers can reduce production costs and make it easier to machine high tolerance parts compared with making the part on two different machines.

There are many configurations of mill-turn lathes, including machines with two tool turrets, two spindles, and even 5-axis milling. These can be challenging to program manually, but the CNC control includes functions to make programming easier. Many CAD/CAM systems support mill-turn centers, though the more exotic the machine configuration, the more difficult it may be to properly simulate and automatically generate edit-free G-code files using CAD/CAM.

8.4 Imaginary Tool Tip

The corners of lathe tools are radii. The imaginary tool tip is where vertical and horizontal lines tangent to the cutting edge of the tool intersect. This point is found by touching off the tool in the Z and X directions on the part or using a tool probe.

Because the imaginary tool tip can be found with great precision, it is used to control the tool. That is, all the Z-X coordinates in the G-code program are in reference to this point. Tool positions are easy to calculate parallel to the machine axes, but more complex for arcs and chamfers.

However, since almost everyone programs using CAD/CAM software or an on-control programming utility, you won't have to calculate complex lathe tool paths. The programmer inputs the part profile geometry, and the control or software does all the calculations. Let the CAD/CAM software or control do this work.

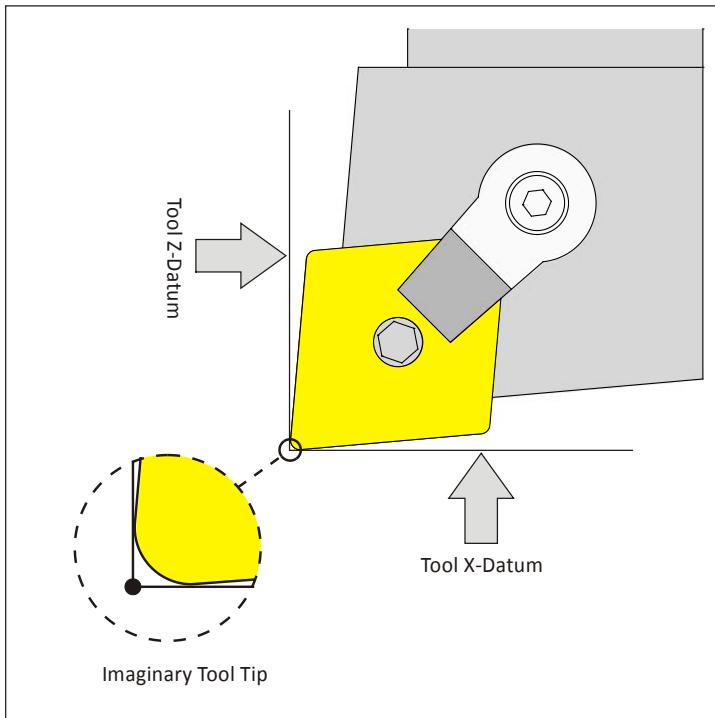


Figure 7: Imaginary Tool Tip

When programming a lathe, think in terms of motion of the imaginary tool tip. As shown in Figure 8, more positive Z-values move the tool to the right. More positive X-values move the tool away from the part.

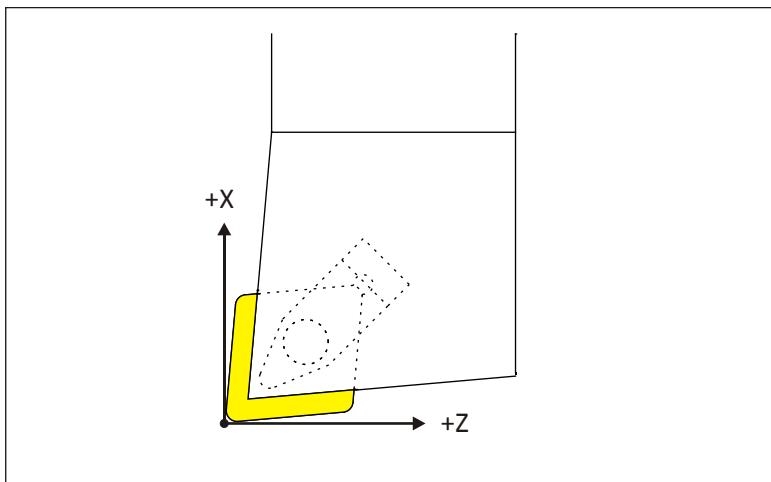


Figure 8: Tool Motion

8.5 - Part Datum

The part datum is usually set as the center-face of the finished part, as shown in Figure 8 below.

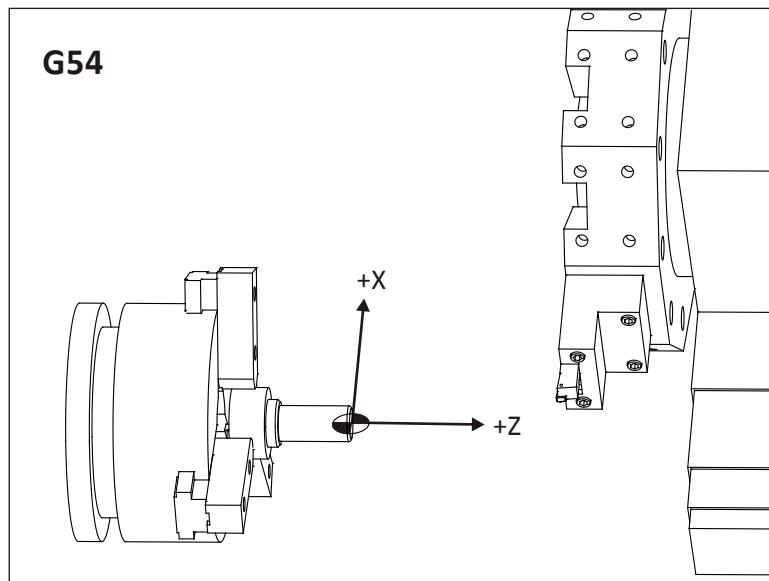


Figure 9: Part Datum

Lathe tools have two tool offsets: Z and X, which is the distance from the imaginary tool tip at home position to the part datum. There are two ways to set lathe tool offsets. The first involves making a small cut (try cut) on the OD and face of the stock material. The diameter is measured and entered in the control for the X-axis offset. Subsequent tool offsets are found in reference to these faces.

Many lathes now include a tool probe as standard or optional equipment. Please refer to your machine documentation for specific instructions about your machine and its options.

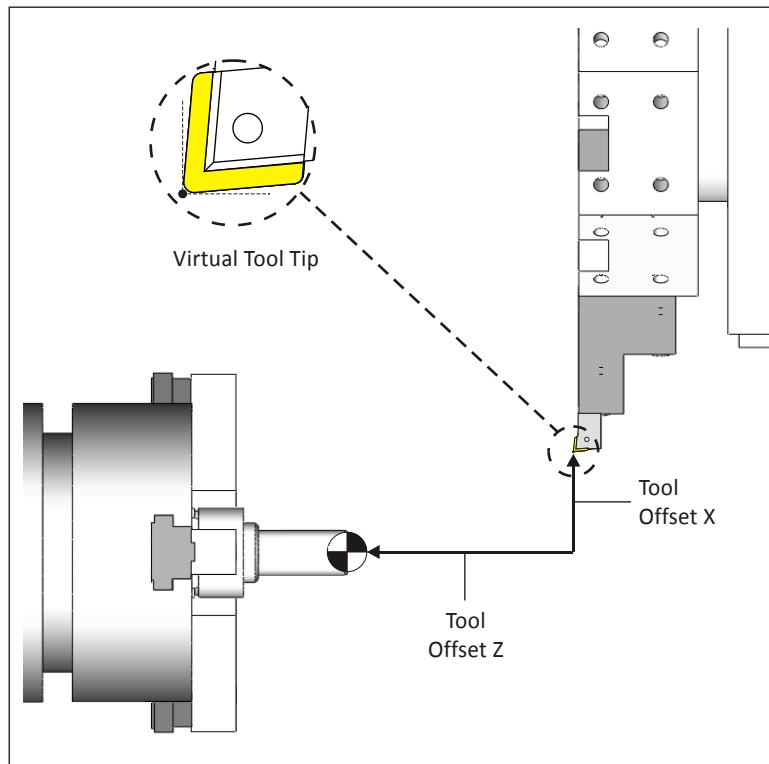


Figure 10: Tool Offsets

8.6 – Tools and Tool Holders

While lathes use some of the same tools that mills use, including spot drills, drills, and taps, most turning is done using carbide inserts. Inserts are gripped in holders, which in turn are bolted to the lathe turret (see Fig. 4). Figure 11 shows a typical insert-holder combination. This is a left-handed holder, because the tool cutting edge points to the left when viewing the holder from the top as shown in Figure 12.

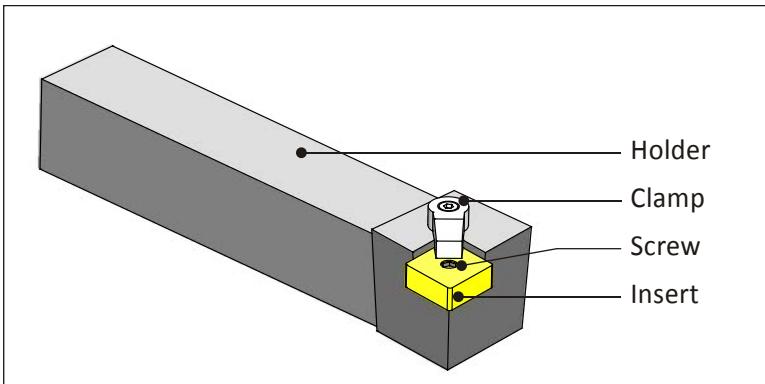


Figure 11: Typical Lathe Tool Holder

Carbide inserts employ highly engineered composite structures, coatings, and geometry features to achieve great accuracy and high material removal rates. Some inserts can be indexed to use other edges when one becomes worn. Inserts are quickly and easily replaced at the machine.

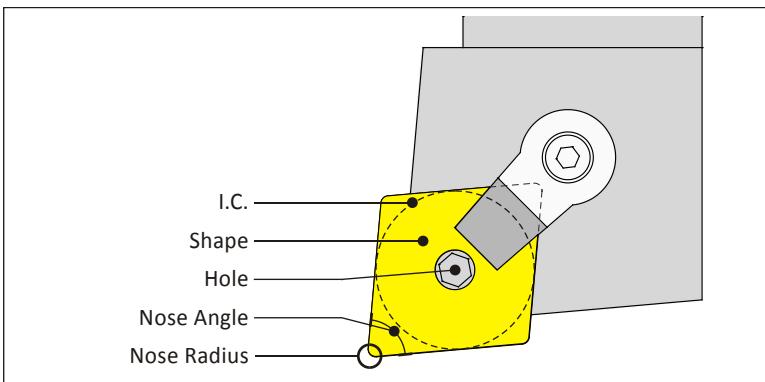


Figure 12: Insert Terms

Chip Breaker

A chip breaker is a feature in the face of the insert that disrupts the flow of chips such that they break into short segments, rather than forming a long, stringy chip.

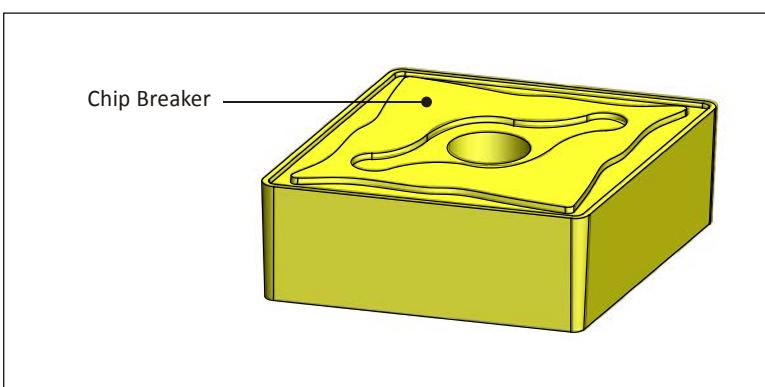


Figure 14: Chip Breaker

Relief Angle

Most inserts have drafted faces on the walls. This is called Relief Angle. Relief prevents the walls of the insert from rubbing against the part.

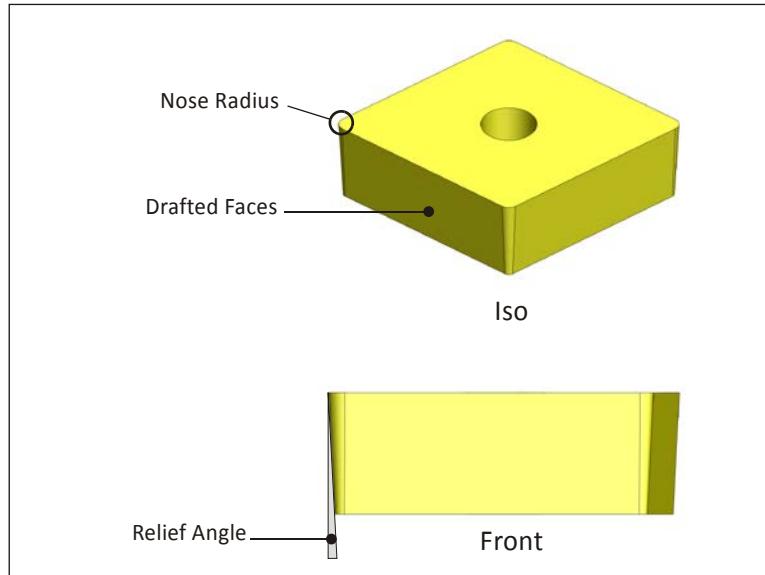


Figure 13: Relief Angle

Tool Cutting Angles

The edge of the tool in the cut direction forms an angle with a line perpendicular to the cut direction. This is called Side Cutting Angle. The angle formed by the trailing edge and parallel to the cut direction is called the End Cutting Angle.

The purpose of these angles is to provide proper clearance between the tool and work piece. For example, the 80 degree insert shown in Figure 16 is rigid and has enough side and end cutting angle for facing and rough turning operations. However, complex contours may require a 55 or 30 degree insert to provide tool side and end clearance for the tool and holder. Very steep or vertical walls may require a round or slot tool to carve.

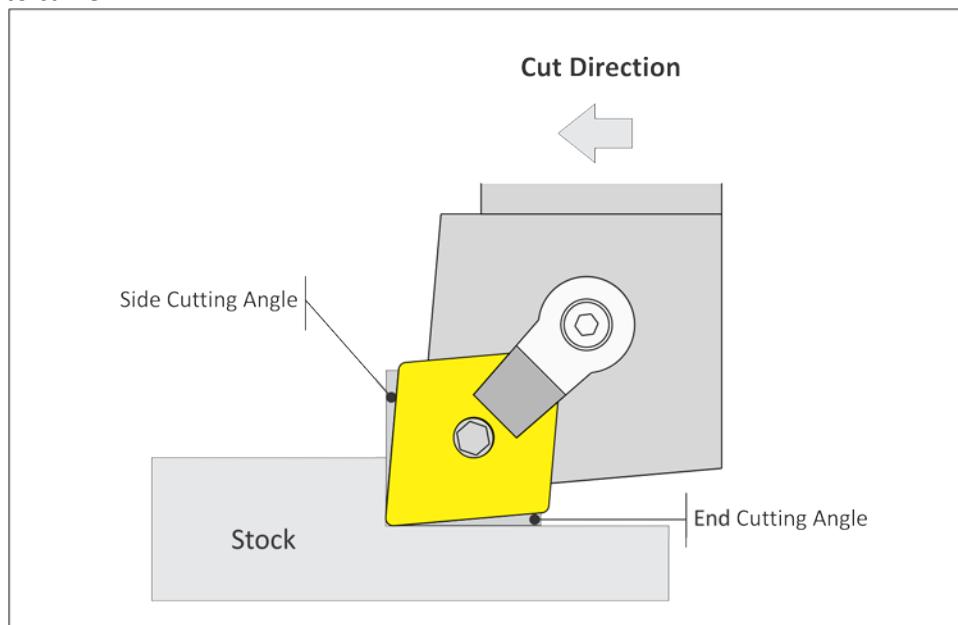


Figure 16: Cutting Angles

Rake Angle

Rake angle is set by the tool holder. Rake angle helps control the direction of the chip and cutting pressure. Angle is measured from face of the insert to the Z-X plane of the machine.

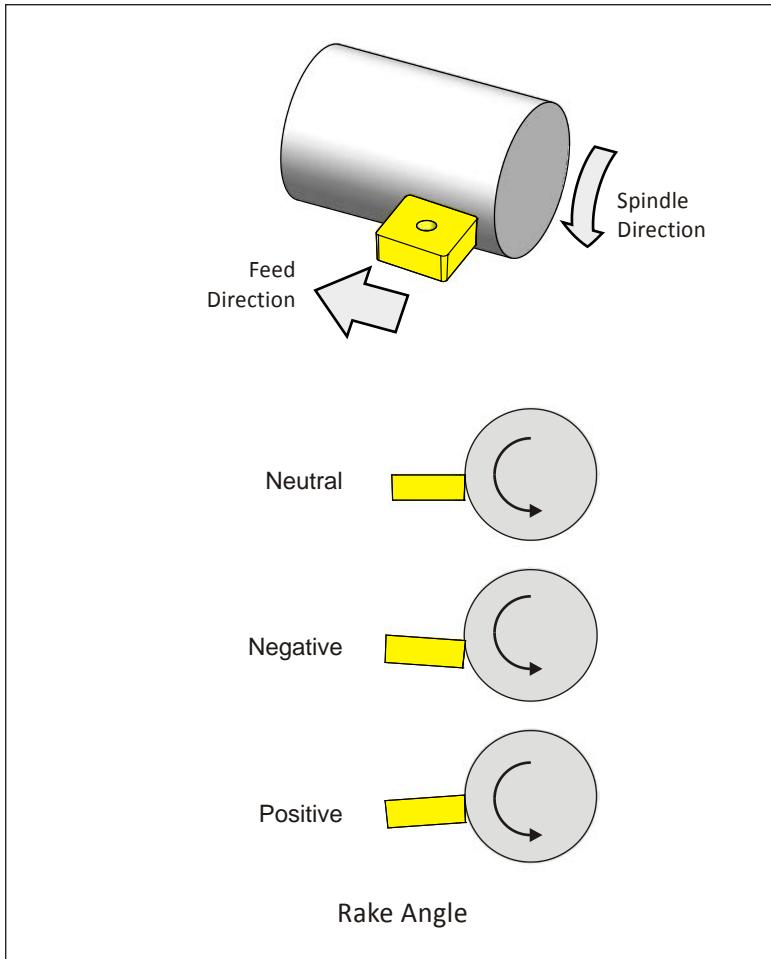


Figure 17: Rake Angle

8.7 Insert Designations

Carbide inserts use a coding system of numbers and letters to describe their shape, dimensions, and important parameters. For example, the designation of the insert shown in Figures 11-18 is a **CNMG-433**.

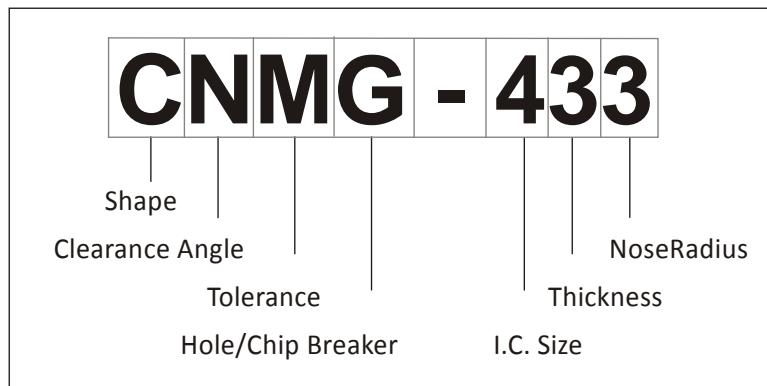


Figure 15: Carbide Insert Designation

Shape (CNMG-433)

There are at least 18 different shapes of carbide inserts. The most commonly used are shown in Table 1 with their letter designation. The angle in this designation refers to the included nose angle at the cutting radius of the tool.

Shape		
T	Triangle	
S	Square	
C	80 degree diamond	
D	55 degree diamond	
V	35 degree diamond	
R	Round	

Table 1: Common Insert Shape Codes

Clearance Angle (CNMG-433)

Clearance angle is the draft on the face(s) of the insert that contact material during machining. More about insert angles a little later.

Designation	Clearance Angle
N	0 Degrees (No Draft)
A	3 Degrees
B	5 Degrees
C	7 Degrees
P	11 Degree

Table 2: Common Insert Clearance Angles

Tolerance (CNMG-433)

This is how much variation is allowed in the dimensional size of the insert. Tolerances described with this parameter include the corner point (nose radius), thickness, and I.C. Typical tolerances are shown in Table 3:

Designation	Cornerpoint	Thickness	I.C.
M	.002-.005	.005	.002-.005
G	.001	.005	.001
E	.001	.001	.001
K	.0005	.001	.002-.005

Table 3: Typical Insert Tolerances (Inch)

Hole/Chip Breaker (CNMG-433)

The hole/chip breaker designation describes both features with one letter. The hole in the insert and tool holder must match. If no letter exists in this field, then the insert does not have a hole to secure it to the holder, and is held by clamp force only.

Designation	Hole Shape	Chipbreaker Type
G	Cylindrical	Single-sided
W	40-60 deg, double c-sink	None
R	None	Single-sided
T	40-60 deg, double c-sink	Single-sided
P	Cylindrical	Hi-double positive
Z	Cylindrical	Hi-double positive

Table 4: Common Insert Hole/Chip Breaker Configurations

I.C. Size (CNMG-433)

Inserts are measured by the diameter of an inscribed circle. I.C.'s range from .0625" to 1.25". Table 5 lists the sizes you are most likely to use.

Designation	Decimal (inch)	Fractional (inch)
3	.375	3/8
4	.500	1 / 2

Table 5: Common Inscribed Circle Sizes

Thickness (CNMG-433)

Insert thickness.

Designation	Decimal (inch)	Fractional (inch)
3	.187	3/16
4	.250	1 / 4

Table 6: Common Inscribed Circle Sizes

Nose Radius (CNMG-433)

Insert cutting nose radius.

Designation	Decimal (inch)	Fractional (inch)
1	.016	1/64
2	.031	1/32
3	.047	3/64

Table 7: Common Inscribed

The insert shapes, sizes, and designations in these tables are just of few of what is available. Any lathe tool catalog or manufacturers web site will show many more.

It is not important to memorize every tool shape or designation scheme. It is important to know insert terms and specifications to understand insert recommendations from the tool representative or technical resource to select the correct insert for the application.

8.8 Lathe Tool Types

The following is a list of the most common lathe-specific tool types. While these are the most common types you are likely to use, they represent only a small number of the tools that are available.

Note: For illustration purposes, the tools shown on the following pages are left-handed. Upper turret machines mostly use right-handed tools.

Face/Turn

For facing and rough turning, use a more rigid tool such as a round, square, or 80 degree diamond. Finishing may require a more versatile tool, such as a 55 or 35 degree diamond. These provide more side and end cutting angle relief to reach and contour part details. Inserts must match the tool holder, and that means the right type, size, shape, and clamping feature(s).

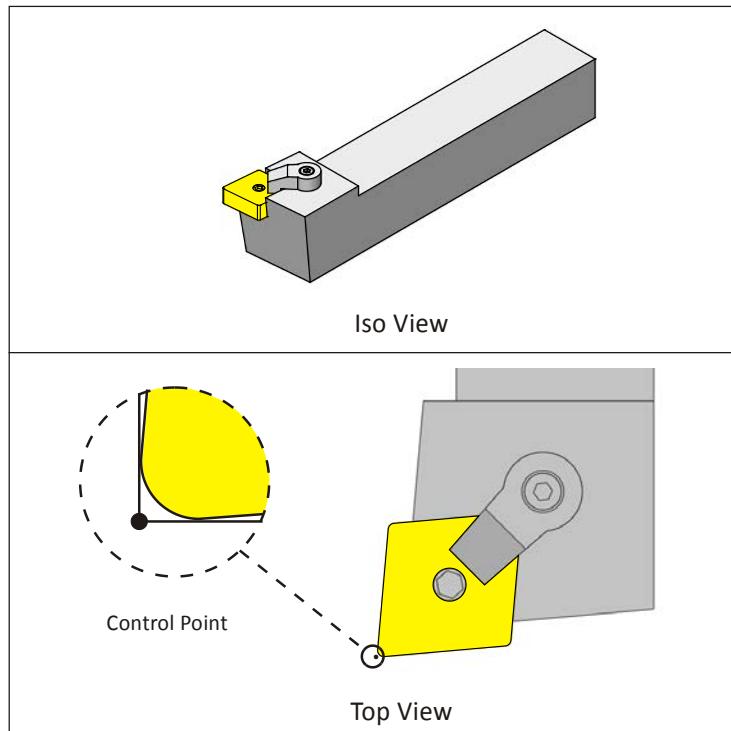


Figure 18: Face/Turn Tool (Left Hand)

Groove

Groove tools are classified in part by their width and corner radii. Though used mostly for making groove features, such as O-ring or snap-ring cuts, newer generations of these tools can be used for rough and finish contouring operations. While not the best choice for all roughing and finishing, they work well in areas where a diamond or other shape cannot easily fit.

Besides there being many types of groove tools, there are many types of holders, depending mostly on the cut direction for the tool. For example, there are groove holders for OD, ID, and Face grooves, and these are all available in Left Hand or Right Hand models.

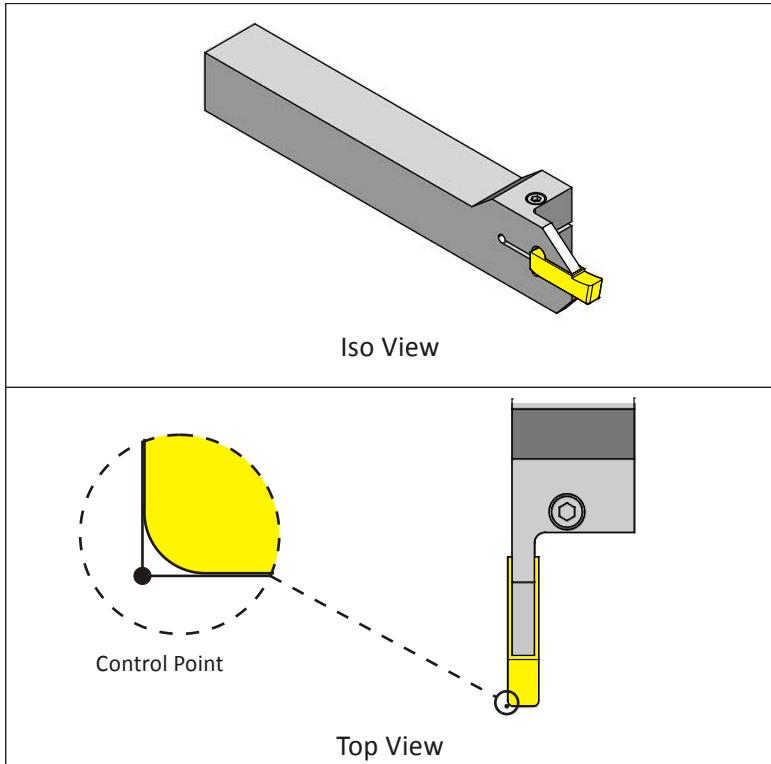


Figure 19: Groove Tool (Left Hand)

Bore

Precision holes are often finished with a boring tool. Boring bar tools are mounted parallel to the machine spindle. They require a hole in the part large enough to allow the bar to safely enter and exit the bore.

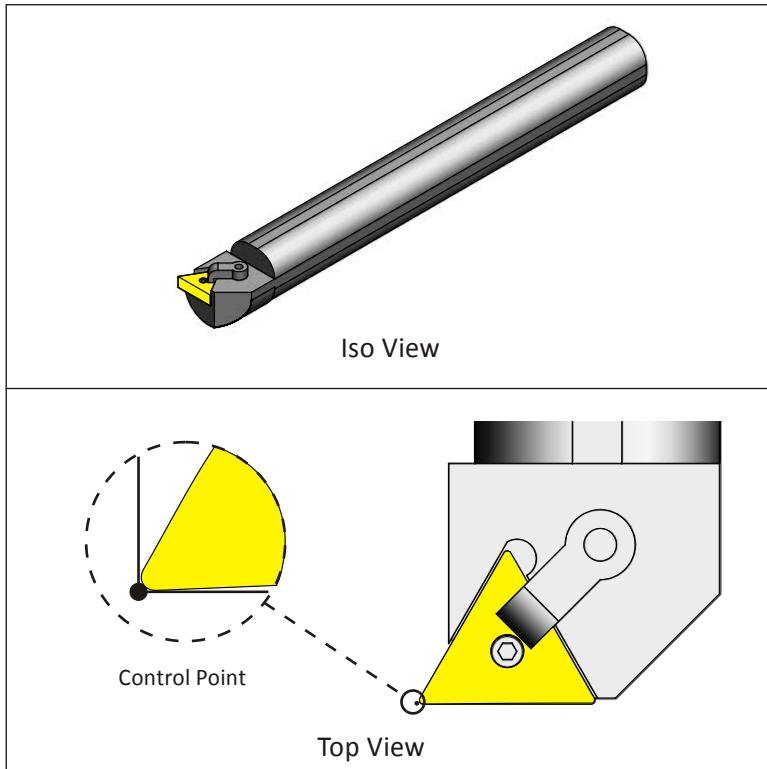


Figure 20: Boring Bar (Left Hand)

Thread

Tapped holes at the center of part, up to about one inch diameter, can be made using a form or cutting tap, just like on a mill. Larger ID threads and all ID threads use a thread insert.

Thread tools are set to the tip of the thread point in Z and X. Z is set by touching off on the edge of the part, and then taking into account the distance from this edge to the tip of the thread point, a distance included in the insert documentation.

Often a thread gage is used to check threads, and the X-offset for the thread tool adjusted to achieve the proper size and fit (Thread Class).

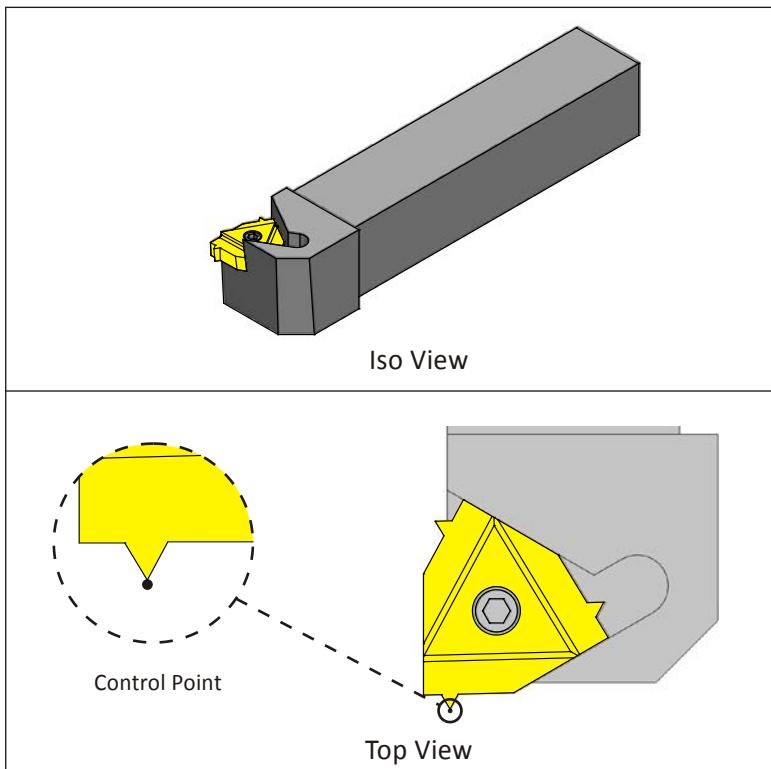


Figure 21: OD Thread Tool (Left Handed)

Cutoff

Once the part is finished, it is usually parted, or cut off from the stock. A cutoff tool is a special kind of groove tool that is designed to take deeper cuts. Cutoff tools are classified in part by their width and maximum cutting depth.

The blade shape of the cutoff tool allows it to cut deeper into the material than a groove tool. This shape does limit the side forces the tool can withstand.

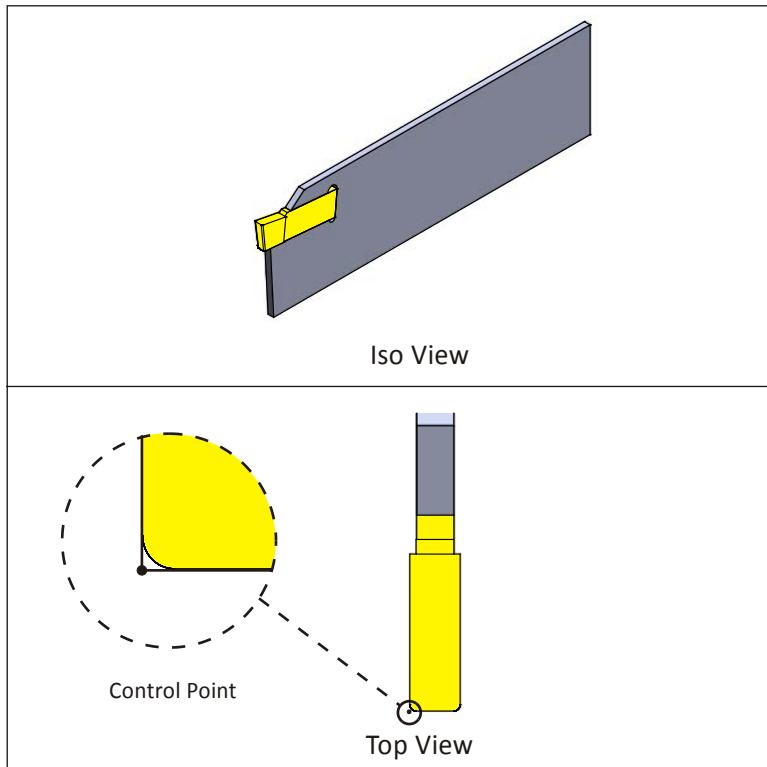


Figure 22: Cutoff Tool

8.9 Cutting Speeds and Feeds

Speeds and feeds are calculated based on the maximum work diameter for the tool. Use the same speed formulas as for mills.

$$\text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) = \frac{\text{SFM} \left(\frac{\text{ft}}{\text{min}} \right) \times 3.82}{\text{Dia}(\text{in})}$$

Figure 23: Speed Formula (Simplified)

Speed is the machine spindle RPM (how fast the part turns).

SFM is the Surface Feet per Minute based on the insert manufacturers' recommendation for the material and cutting conditions.

3.82 is a constant that relates the surface feet per minute to work piece diameter.

Dia is the diameter of the tool when it first engages the material.

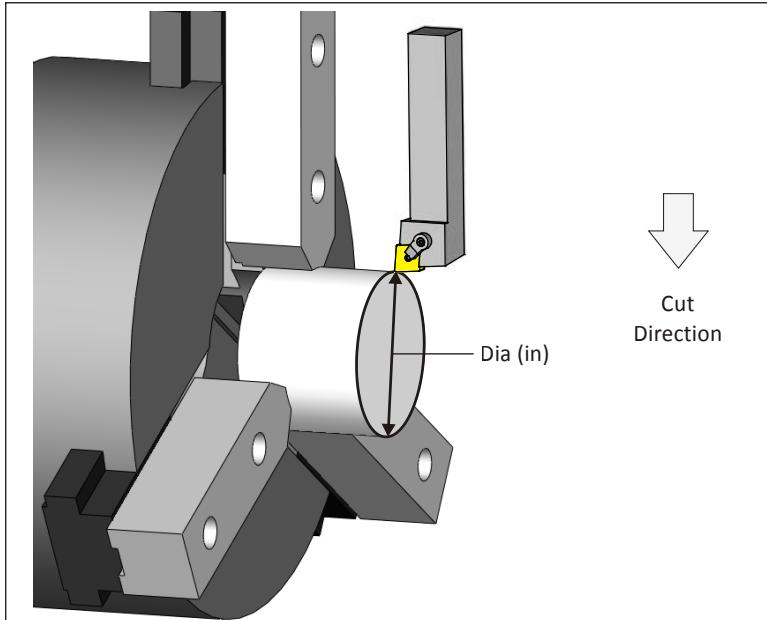


Figure 24: Facing Tool Engaging Material

Speed/Feed Example

Problem:

Calculate the cutting speed for a facing operation given the following values:

Parameter	Value
Part Diameter	2.0in
SFM	500ft/min

Table 8: Drill Speed/Feed Example

Solution:

$$\text{Speed}(\frac{\text{rev}}{\text{min}}) = \frac{\left(\text{SFM}(\frac{\text{ft}}{\text{min}}) \times 3.82 \right)}{\text{Dia}(in)}$$

$$\text{Speed}(\frac{\text{rev}}{\text{min}}) = \frac{500(\frac{\text{ft}}{\text{min}}) \times 3.82}{2.0\text{in}}$$

$$\text{Speed}(\frac{\text{rev}}{\text{min}}) = 955 \frac{\text{rev}}{\text{min}}$$

Constant Surface Speed (CSS)

Notice that as the tool moves down the face of the part, the diameter where the cutting edge contacts the part gets smaller.

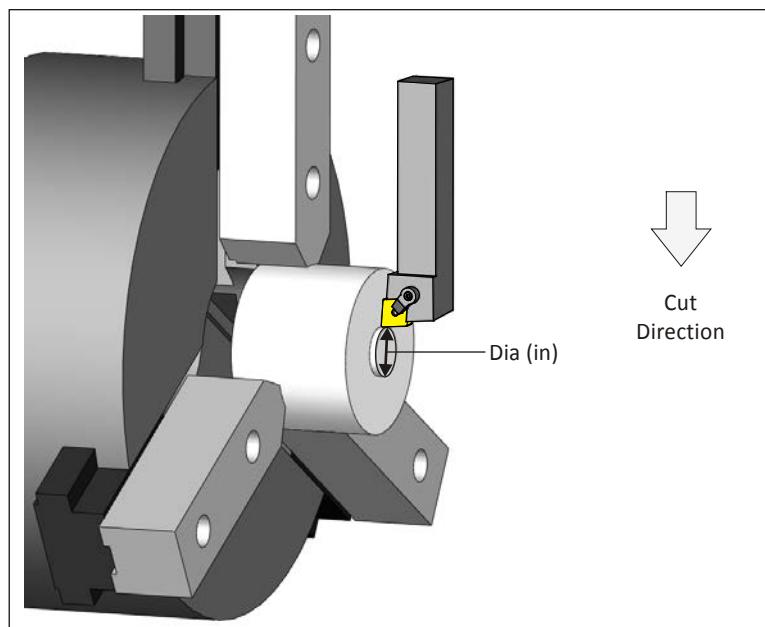


Figure 25: Decreasing Cut Diameter As Tool Nears Center

To maintain a constant rate of material removal as the cutting diameter decreases, most CNC machines automatically speed up the spindle, based on how far the tool moves towards center. This constantly variable spindle control is called Constant Surface Speed (CSS) mode. It is commanded on most machines using G96 to activate, and G97 to de-activate.

G96 is preceded by some important codes, as shown in the following example:

Block	Description
G50 S2000	G50 sets the maximum spindle speed for this tool.
G97 S1200 T101 M03	G97 cancels CSS so the spindle speed does not change as the turret rapids to position.
G00 X2.1 Z0.	Tool moves to start of cut.
M08	Coolant ON.
G96 S1200	CSS mode on, Spindle begins at 1200 RPM.
G01 X-.05	Face part. As tool moves from a diameter of 2.1 inches to X-.05, the spindle will increase speed up to the limit of S2000 RPM.

Table 9: Lathe Program Using CSS

When invoked, you will hear the lathe spindle increase as the tool moves from the perimeter of the cut to the part center. The G50 command is important because it keeps the spindle from over-speeding.

Warning: All lathe chucks have speed limits based on the weight of the part and how far it extends from the face of the chuck. Do not exceed these limits. Ensure chuck clamping pressure is sufficient to grip the part based on cutting forces. Use G50 when using CSS mode to limit the speed of the spindle.

CSS does not apply where the tool does not change its position along X. For example, don't use CSS mode for drilling or tapping on part centerline.

Feed is based on the speed and manufacturer' recommended Feed Per Revolution (FPR) of the tool.

Cutting Feeds Example

Problem: Calculate the cutting feed for tool with the following parameters.

Parameter	Value
Speed	955 rpm
IPR	.015 in/rev

Table 10: Lathe Feed Example

Solution:

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = \text{Speed} \left(\frac{\text{rev}}{\text{min}} \right) \times \text{IPR} \left(\frac{\text{in}}{\text{rev}} \right)$$

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = 955 \left(\frac{\text{rev}}{\text{min}} \right) \times .015 \left(\frac{\text{in}}{\text{rev}} \right)$$

$$\text{Feed} \left(\frac{\text{in}}{\text{min}} \right) = 14.3 \left(\frac{\text{in}}{\text{min}} \right)$$

8.10 Lathe Setup & Programming Example

The following turned part is used on the following pages to introduce lathe tool paths. This example deals with processes in general, so material and dimensions are not important now.

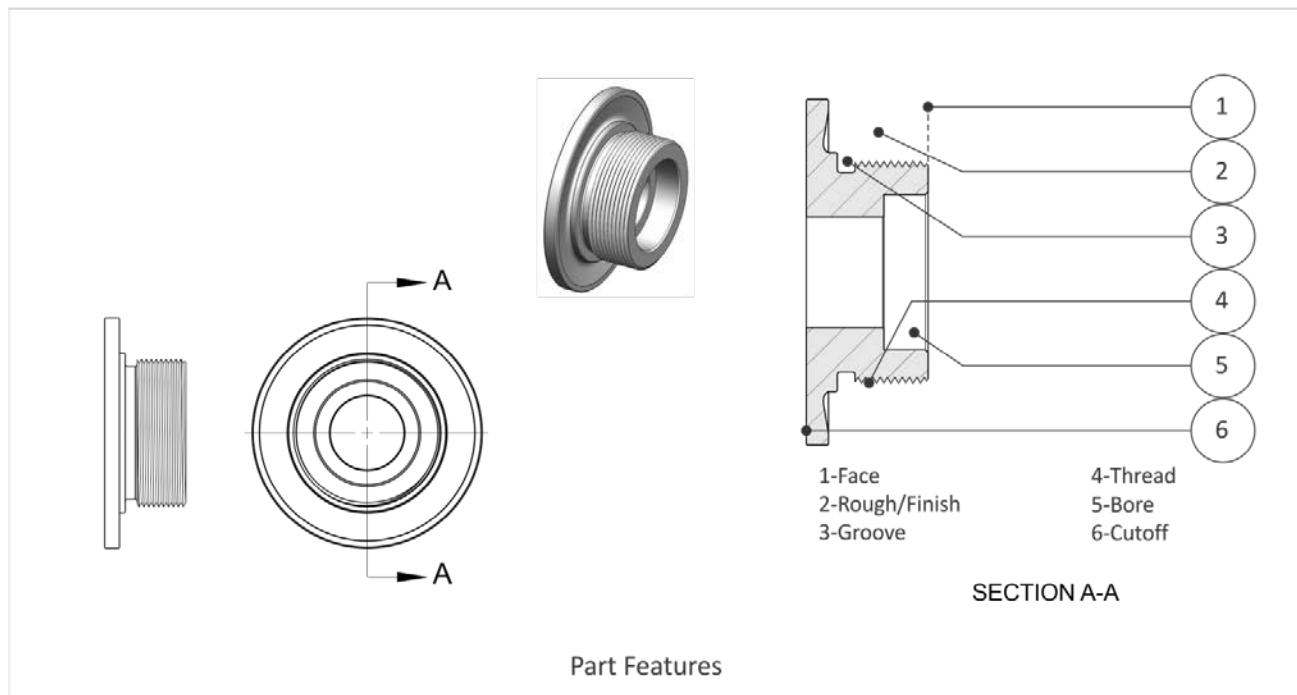


Figure 26: Example Lathe Part and Features

1 – Face

Facing is usually the first lathe operation. It provides a known location to set the Z-offset of other tools and a flat surface for face drilling. Because sawed stock can vary in length, it is a common practice to take at least one roughing and finish facing pass.

2 – Rough/Finish

The goal of lathe roughing is to remove excess material as efficiently as possible, ideally leaving a constant thickness of material for the finish tool. Finish tool paths determine the final size and surface finish.

3 – Groove

Grooves have many configurations, but most have straight walls and radii at the bottom and top that is formed with the groove tool.

4 – Thread

Threads on a lathe are usually machined with a special thread tool. Small diameter threads through the center of the part may be cut with a tap, just like on a mill.

5 – Bore

Clearance holes in the bore may be drilled. Small bore holes may be reamed, but are usually finished with a boring bar tool.

6 – Cutoff

The final operation is the cutoff. This may also put a chamfer or radius on the outside edge of the part. Once cut away from the stock, the part is allowed to simply fall into the bottom of the machine, though many

lathes include a part catcher chute that extends during cutoff. The part falls onto this chute and rolls into cage for the operator to retrieve.

Setup

The part is gripped in hard jaws, soft jaws, or collet. Part datum is usually the center-face of the finished part. Provide enough excess stock to securely grip the part. The back side of the part should be as close to the jaws as possible while providing clearance for the cutoff tool.

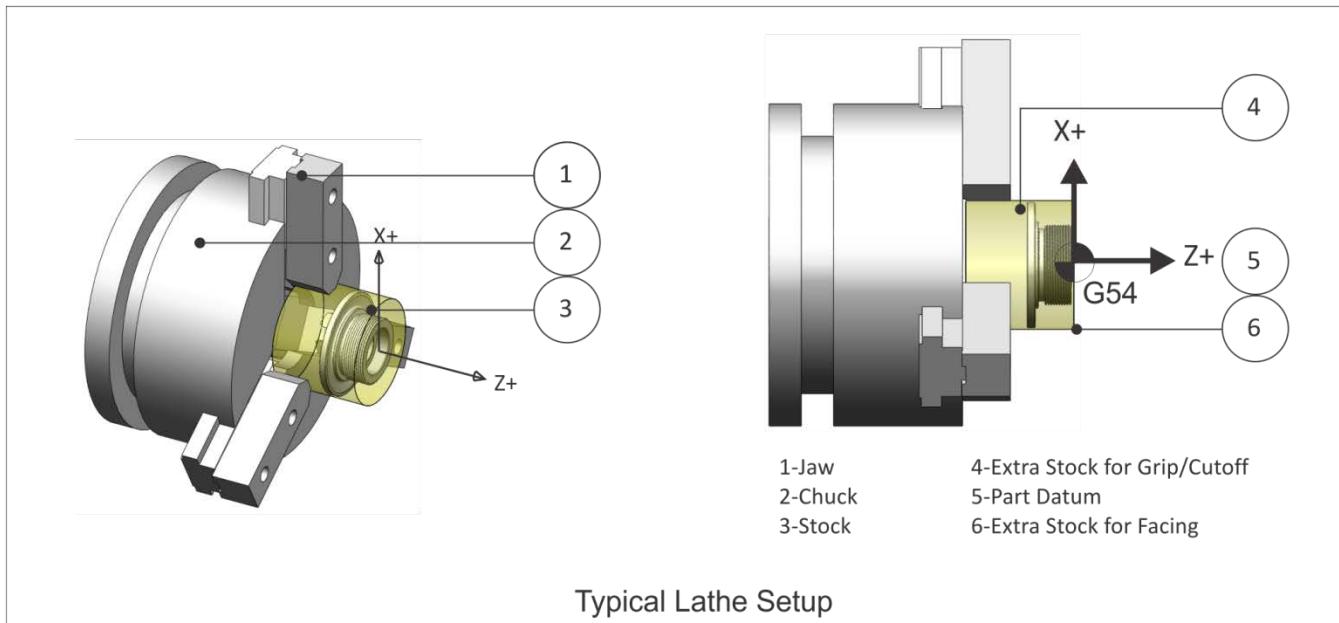


Figure 27: Typical Lathe Setup

1 – Jaws

If using soft ways, machine them to fit the OD of the part. Jaws must have sufficient clamping pressure to hold the part. There is a limit to how far the jaws can be safely extended. This information is found in the chuck documentation.

2–Chuck

The chuck secures the jaws. A hole in the chuck allows bar material to extend through the chuck. This is useful for production runs. After a part is cut away, the jaws are opened, the stock drawn out further, and then the jaws are closed. This saves cutting the stock to length and wasted material.

Warning: There is a limit to how far the stock can extend out of the chuck without supporting the other end. Check your machine and chuck documentation for this and other operating limits.

3–Stock

Stock needs to be over-sized in Z and X so the face and finish tools remove material all around, including the cutoff.

4–Grip/Cutoff Stock

Ensure the back side of the part is far enough away from the jaws to provide cutoff tool clearance. Lathes are extremely powerful and any collision between the tool and rotating jaws is dangerous.

5– Part Datum

The part datum for lathe parts is usually the center-face of the finished part.

6–Extra Stock for Facing

Sawed stock is uneven and has a poor finish. Ensure the stock extends slightly beyond the finished face of the part so the face pass removes material.

Face

Facing is usually the first operation of any turning job.

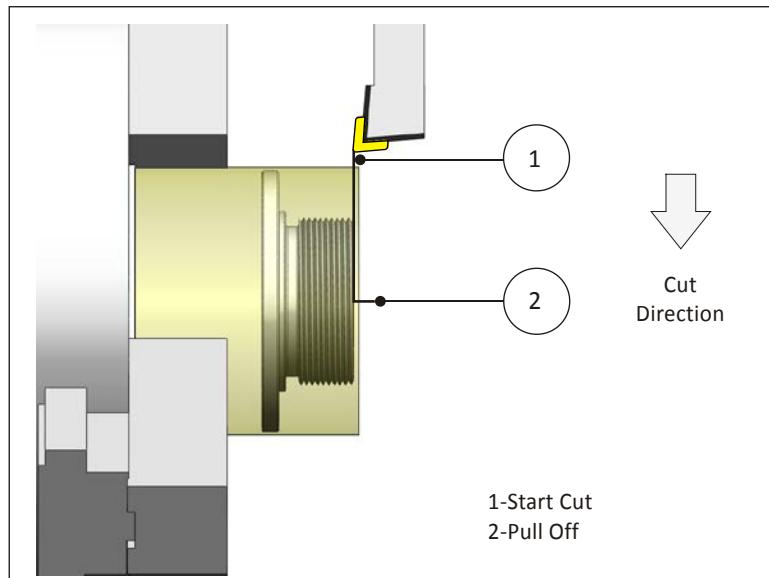


Figure 28: Face Toolpath

Rules for Facing

- Start away from the OD of the stock, keeping in mind that rough stock diameter varies.
- Face straight down until the contact point of the tool is at X0 and then pull away.
- Consider taking one rough and one finish pass.
- Use a rigid tool, such as a 80 degree diamond.

Rough

Rough the part leaving stock in Z and X for finish paths.

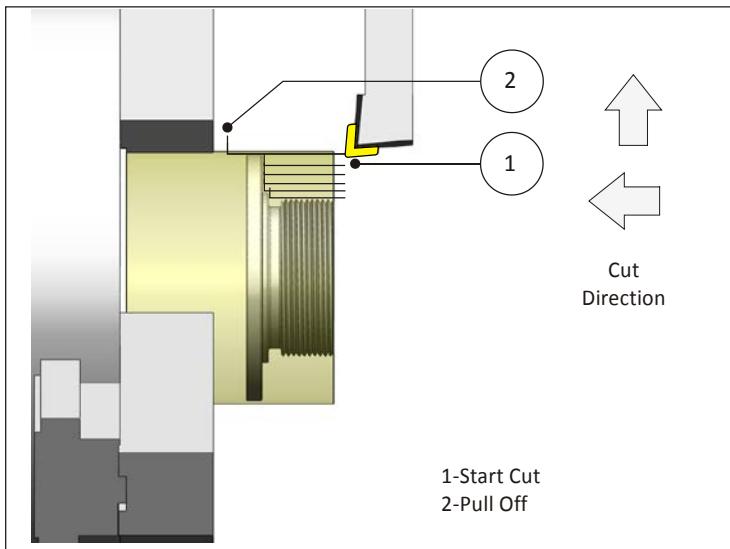


Figure 29: Roughing

Rules for Roughing

- Start away from the face of the part.
- A common practice is to use the same tool for both facing and OD roughing when possible.
- Leave a constant thickness of material over the part for the finishing operation.
- Skip over grooves and other features that will be rough or finished with other tools and operations.
- Consider moving the tool vertically at the end of each path to remove scallops.
- Extend the path past the back of the part for cutoff tool clearance.

Finish

Finish the part. Choose the largest and most rigid tool that will make all the features. Watch the back clearance of the tool to ensure the tool and holder do not collide with the part.

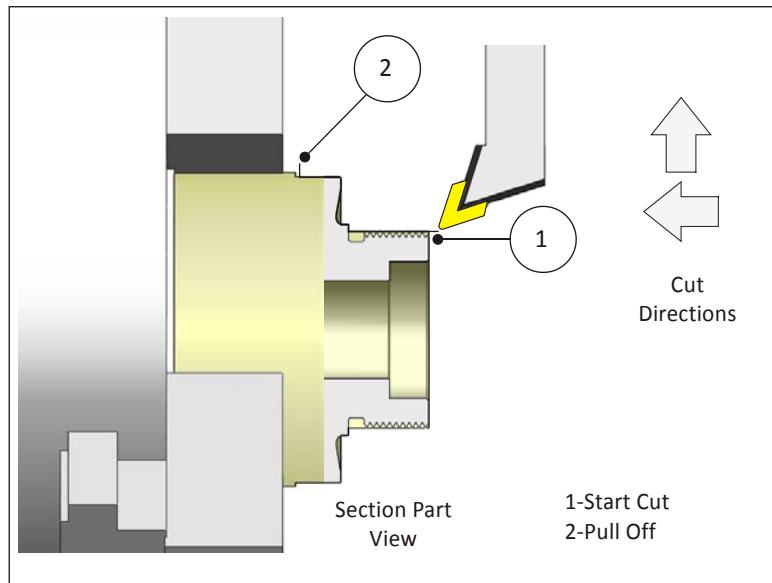


Figure 30: Finish

Rules for Finishing

- Start away from the part face.
- Use a tool, such as a 35 or 55 degree diamond, with sufficient side and end cutting angles to provide tool clearance.
- Use a tool with a nose radius equal, or preferably smaller, than the smaller ID radius in the profile.
- Consider taking a pre-finish pass first so the final pass removes a constant thickness of material.
- Skip over features, such as grooves, that will be machined using other tools and operations.
- For threads, finish at the major diameter of the thread.

Groove

Most grooves have radii at the top and bottom. Groove tool paths are actually quite complex. The following sequence is often used to create a groove that is accurate and with no tool marks.

A common groove operation is shown in the Details view in Figure 31: A) The tool is usually plunged near the center of the groove to rough. B) Other plunge moves as required to remove excess material. C) One contouring path, starting outside and moving inward, creating radii and contour. D) One contouring path, starting from the other side and moving inward finishing the other side.

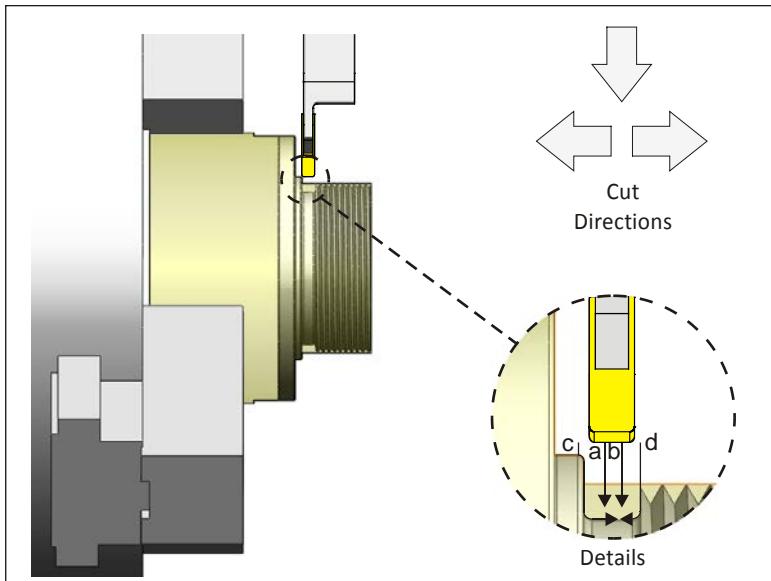


Figure 31: Groove

Rules for Grooving

- Use a groove tool narrower than the groove and with a radius equal or less than the finished radius.
- Use CAD/CAM or lathe control groove cycles to program the grooves.

Thread

Lathes are capable of creating any form thread, including tapered pipe threads, OD, ID, and multi-start. Most threading on a lathe is done with an insert. Several roughing passes that decrease in depth with each cut are followed by one or more finish passes that remove a small amount of material to form a thread that is accurate and smooth.

The motion of the tool and spindle are coordinated by the CNC control so that each time the tool begins the cut pass at the same rotational position.

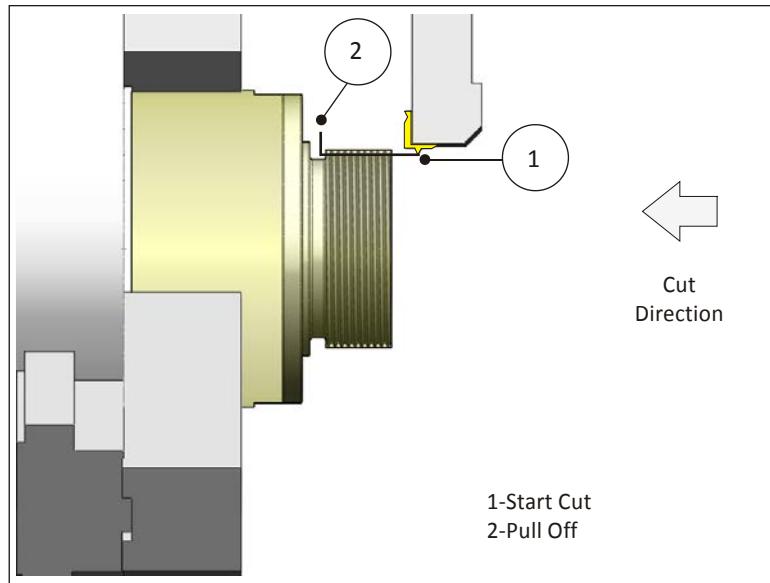


Figure 32: Thread

Rules for Threading

- Use the insert manufacturers' recommendations for thread parameters.
- Use the CAD/CAM software or lathe control thread canned cycles. These work much like a mill drill cycle to program all the motion required for form the thread with just a few blocks of G-code.
- Start well away from the thread to give the spindle time to reach full speed before engaging the thread.

Drill

Lathes support common drill G-code cycles, including G81.

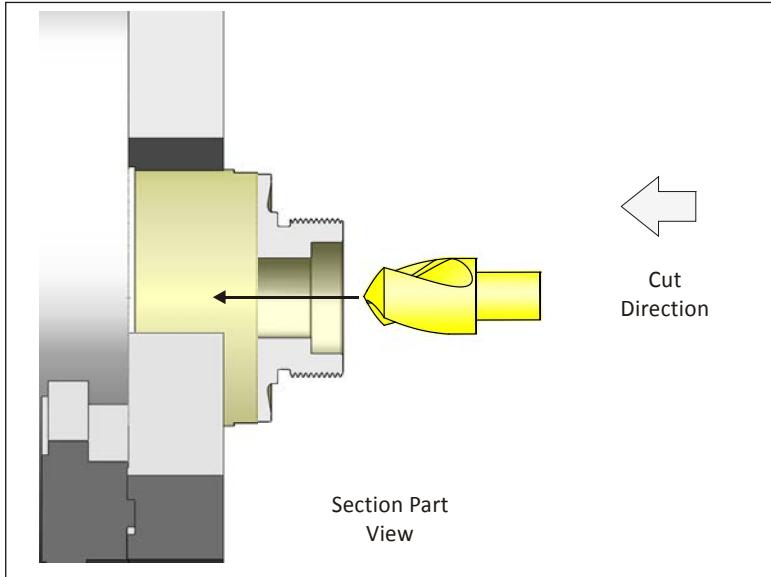


Figure 33: Drill

Rules for Drilling

- Spot drill holes.
- Consider using progressively larger drills to make a large hole.
- Peck drill deeper holes.
- Use a canned cycle when possible. If programming using line and rapid moves, be sure to extract the tool completely from the bore before changing tools.
- Use the shortest drill possible to reduce tool chatter and produce a more accurate hole.

Bore

Bore operations produce precision holes with excellent surface finish.

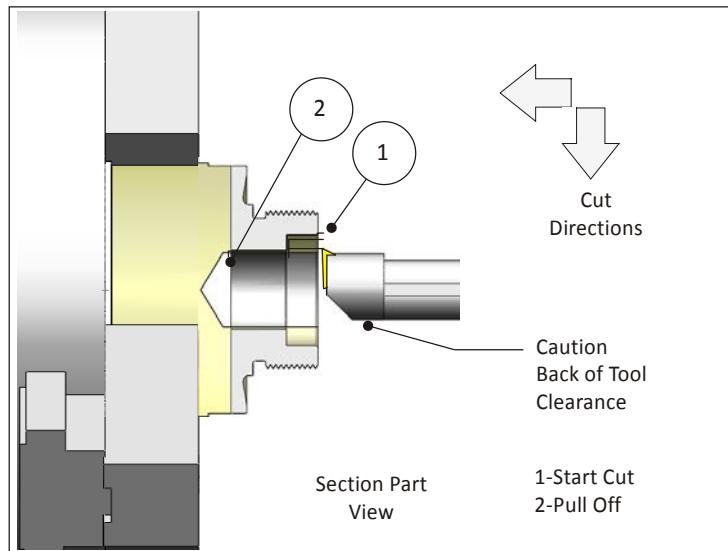


Figure 43: Boring

Rules for Boring:

- Ensure the hole for the boring bar is large enough for tool clearance during all moves, including the lead-out move away from the finished surface.
- Be sure to fully extract the bore bar out of the hole before sending it home.
- Precision holes require the correct amount of stock remaining and proper cutting speeds and feeds.
- Add roughing passes as needed before taking the finish cut to ensure constant material thickness for the finish pass.
- Extend the tool path slightly beyond the back of the part to ensure a clean edge after cut off.
- For precision holes, such as press fits, consider backing off the X-offset several thousandths of an inch on the first part. Machine the hole, check the actual finished bore diameter, and then adjust the offset as needed to produce the precision hole.

Cutoff

This is always the last lathe machining operation. It frees the part from the stock material.

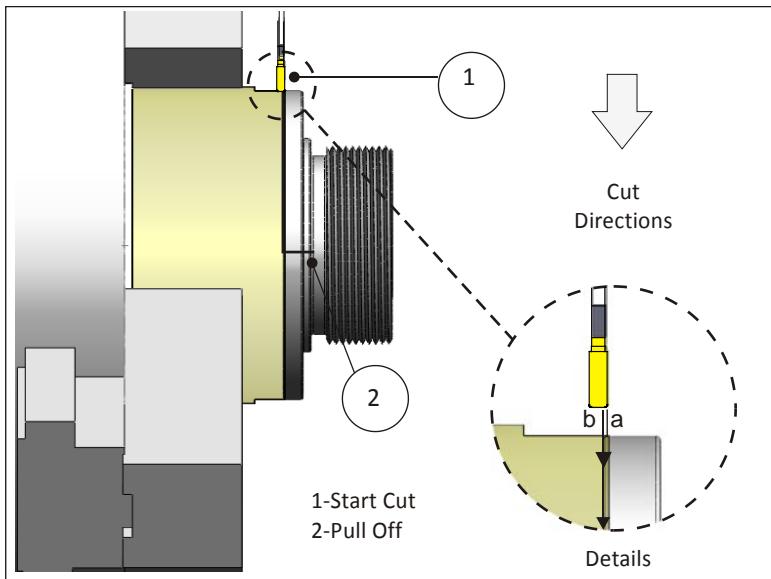


Figure 44: Cutoff

Rules for Cutoff:

- Cutoff operations are much like grooving operations.
- As shown in the detail view above, cutoff usually begins with a cut that forms a radius or chamfer on the back side of the part. The tool then retracts, moves back, and then plunges to cut away the part.
- Consider using a parts catcher to keep the part from falling into the bottom of the machine and being scratched by the cut chips.

Lesson 9

3D Toolpaths

Upon successful completion of this lesson, you will be able to:

- Explain how 3D tool compensation is calculated.
- Describe 3D cut tolerances.
- Identify geometric features common to 3D tool paths.
- Explain the purpose, general parameters, and use of common 3D roughing tool paths.
- Explain the purpose, general parameters, and use of common 3D finishing tool paths.
- Explain the purpose, general parameters, and use of REST mill tool paths.
- Explain the purpose, general parameters, and use of Pencil mill tool paths.

Overview

3D tool paths are used to machine non-prismatic parts such as molds, dies, and organically shaped consumer products. These parts may be composed of hundreds or thousands of faces.

CAD/CAM software creates 3D tool paths by first triangulating the model, as shown in Figure 1 (though the mesh is far finer than shown in this illustration). This mesh is used to calculate the tool path based on tool size and shape.

3D tool paths are calculation intensive in part because of the extensive checking required to ensure the tool does not gouge the part as it moves across the topography of part faces.

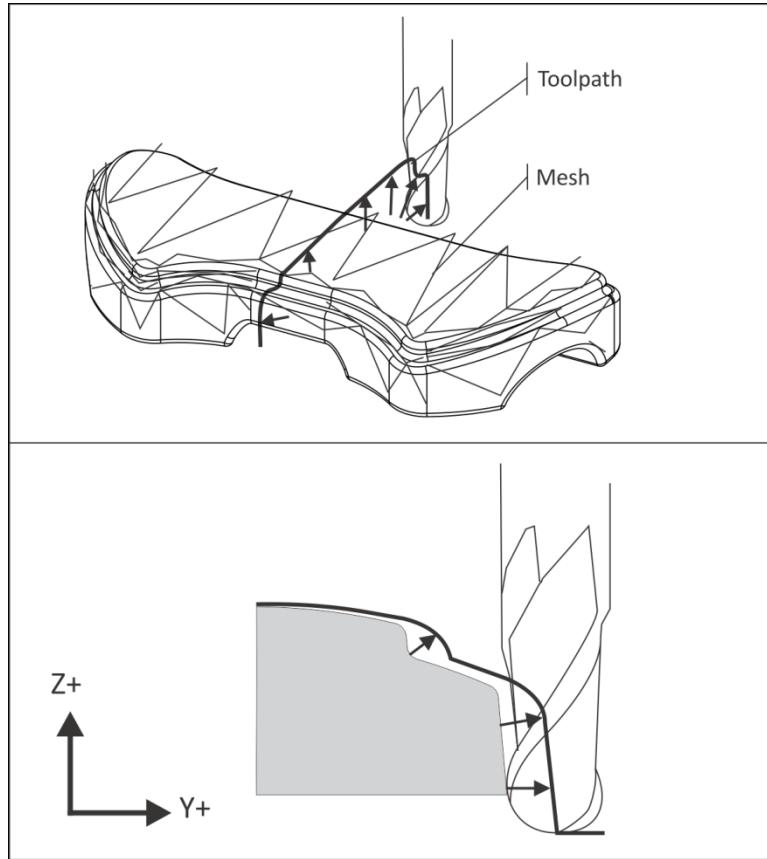


Figure 1: 3D Tool Path Calculation

9.1 – 3D Cutter Compensation

3D paths control the center-tip of the tool. Figure 1 shows and compares how the tool path changes based on tool shape

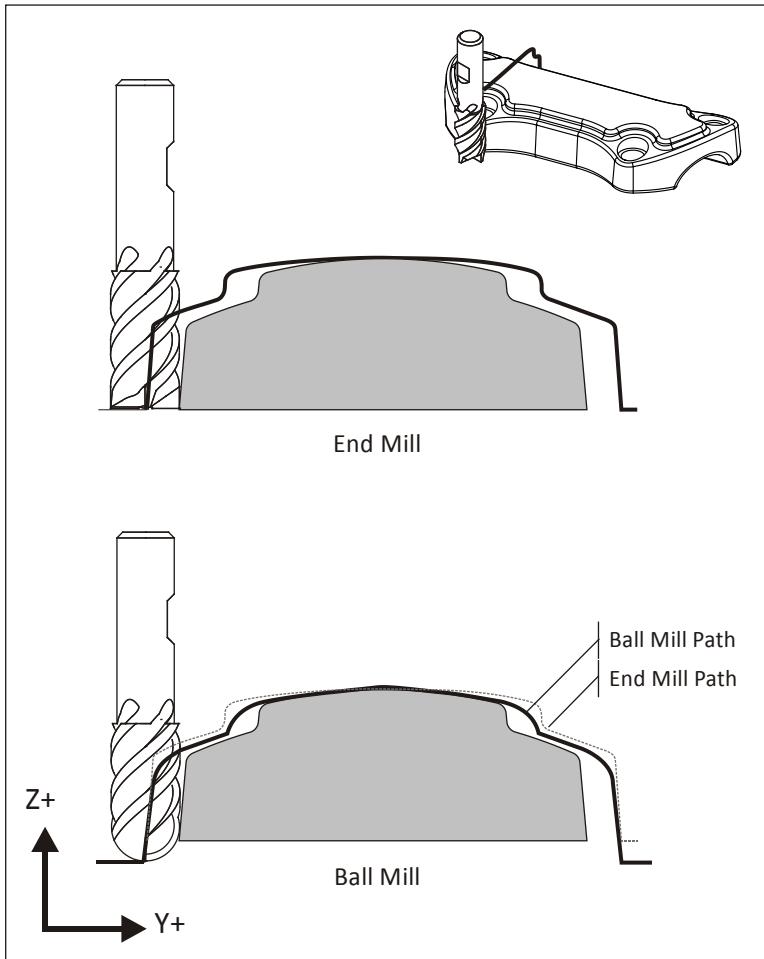


Figure 2: 3D Cutter Compensation

Cutter Diameter Compensation (G41/G42) is not supported for 3D tool paths by most CNC controls. The only way to compensate for a worn 3D finishing tool is to replace it.

9.2 – Tolerances

3D tool paths are much about tradeoffs between quality and cost. As cut tolerances shrink, calculation time, file size, and run times balloon. Excessively fine tolerances may result in hundreds or thousands of blocks of code to move a short distance. This not only takes longer to calculate, it causes problems at the machine. It is therefore important to understand all tolerances involved with 3D tool paths, choose values appropriate to the task, and take actions to reduce CNC program file size while maintaining design intent.

CNC programs with unnecessarily fine tolerances should be avoided. CNC controls can only process a finite number of blocks of code per second. This processing speed, called the **block execution time**, varies between machines. Modern machines may be capable of processing several thousand blocks of code each second while older controls may be limited to less than a hundred.

If the number of blocks per second exceeds the machine capacity to process, a phenomenon known as **data starving** can occur: the control is overwhelmed with data and must pause after each move to wait for the next. This abrupt stopping and starting may happen hundreds of times per second.

Due to the mass of the machine components, the result is an effect similar to alternately pushing the accelerator then brake in a car. The machine may physically shake and shudder, a phenomenon called "bumping". The actual feed rate of the machine may be a fraction of the programmed feed rate.

Data starving causes excessive CNC machine wear, poor surface finish, and excessive program run times. One solution at the machine is to use the machine manual feed rate override to feed rate until the shuddering stops. Some machines allow the control acceleration/deceleration function to be disabled or modified.

Much can be done at the CNC programming stage to avoid or correct this problem. Choose machining tolerances wisely. Use the CAD/CAM software tool path **filtering** function. Filtering works by analyzing the cut path and fitting long lines or tangent arcs where possible to replace short line moves. This may significantly reduce program size (by as much as 90%) while increasing the span of each move.

One consideration when choosing a tool path strategy is how well it will filter. Paths parallel to a CNC work plane (G17/G18/G19) filter far better than paths that are not.

Cut tolerance, illustrated in Figure 3, controls how closely the tool path follows a theoretically perfect path along the surface. Cut tolerance is a plus or minus value, so the total cut tolerance band is twice the tolerance value.

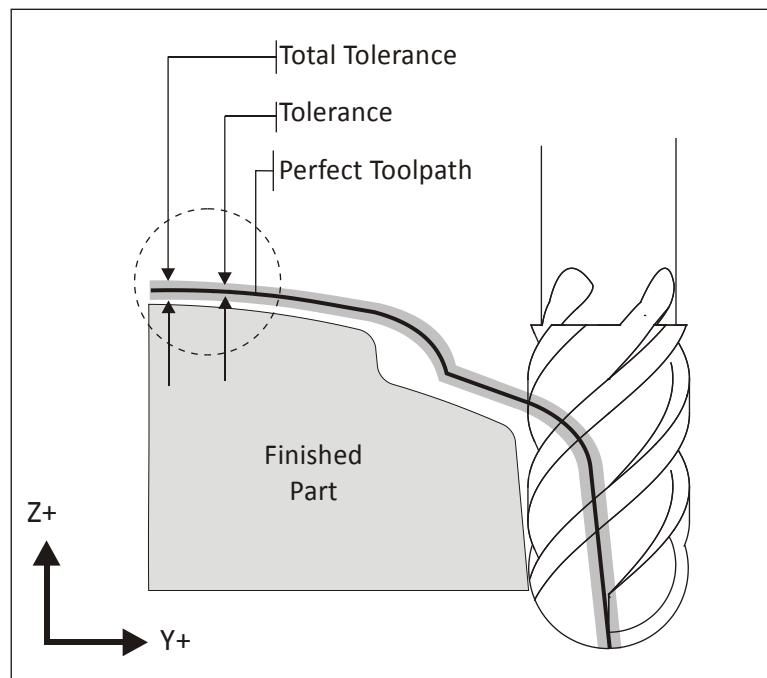


Figure 3: Cut Tolerance

Cut tolerance and filter tolerances are additive. For instance, a cut tolerance of .0005 and filter tolerance of .001 means the tool path can vary from the theoretically perfect path by as much as .0015 inches to either side of the planned path.

9.3 – 3D Toolpath Setups

A typical 3D setup is shown in Figure 4. **Runoff surfaces** are sometimes required to expand the tool paths to the XY extents of the stock, or to cause the tool to continue to machine down in Z along vertical walls.

Holes, fine details, or other features that will be created by subsequent operations may be suppressed or covered with a **Check Surface** to prevent the tool from entering these areas.

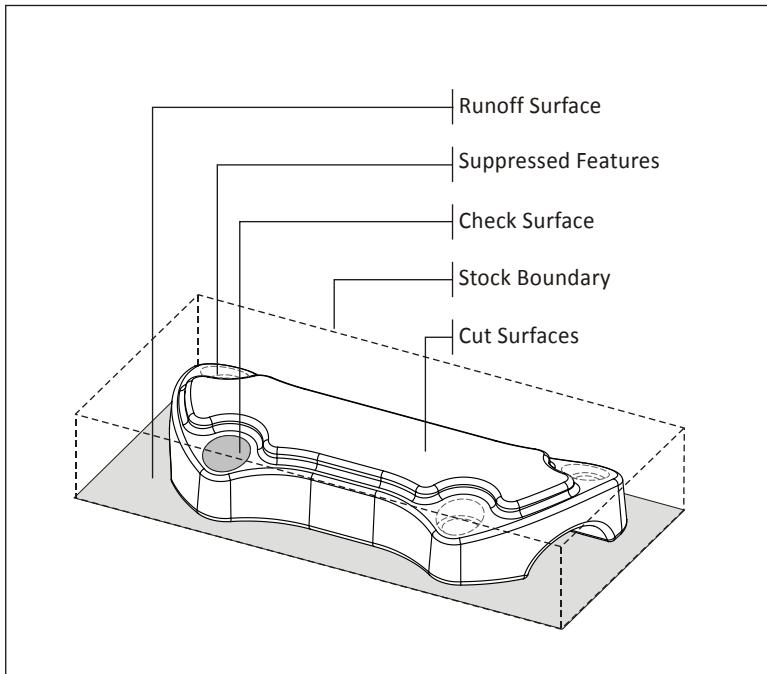


Figure 4: Typical 3D Tool Path Setup

9.4 – 3D Roughing

3D tool paths can be grouped into two broad classifications: Roughing and Finishing. The aim of roughing tool paths is to remove excess material and, ideally, leave a constant thickness of stock all over the part for the finishing operations.

CAM systems include many strategies for roughing but by far the most common is some variation of 3D pocketing. These tool paths work by slicing the part by planes normal to the Z-axis. A boundary is created at each level, offset by the stock allowance, and 2D pocket tool path generated from this boundary. The result is a tiered cake shape as shown by the shaded image in Figure 5.

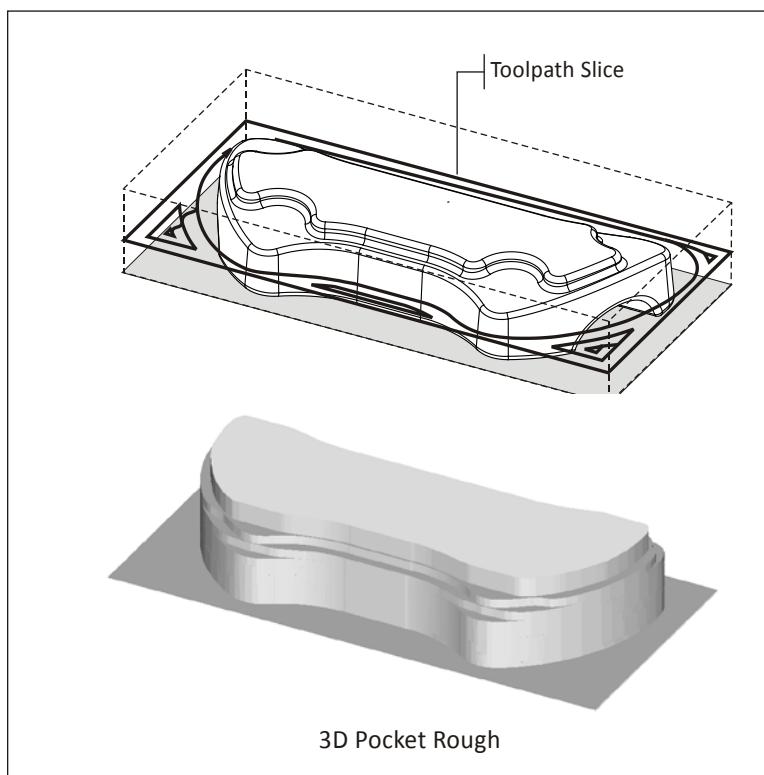


Figure 5: 3D Pocket Rough

9.5 – Parallel Finish

The goal of 3D finishing tool paths is to remove material left by roughing paths and produce a part that meets design requirements for dimensional accuracy and surface finish.

A commonly used finish path is **Parallel**, shown in Figure 6. Parallel gets its name because, when viewed from above, tool paths appear parallel to each other.

Parallel tool paths calculate quickly and are reliable. However, they usually require additional finish passes to clean up heavy scallops (cusps).

Scallops can be seen Figure 7 and are most prominent on the closest wall. Small tool path step over values produce smaller scallops. Notice how scallop height changes depending on the topography of the part.

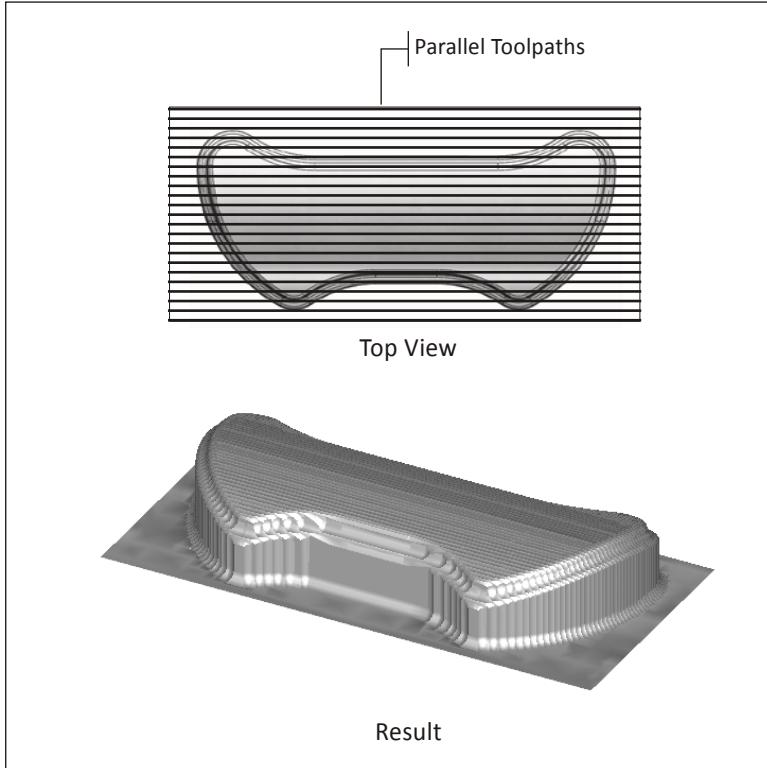


Figure 6: 3D Parallel Tool Path

Parallel tool paths tend to produce large scallops on steep walls roughly parallel to the path direction. As the tool steps to the next pass, the path drops down farther in Z on these walls compared with flat areas of the part. One approach to machining away these scallops is to create an additional parallel finish path rotated 90 degrees to the first. Of course this increases total program run time substantially.

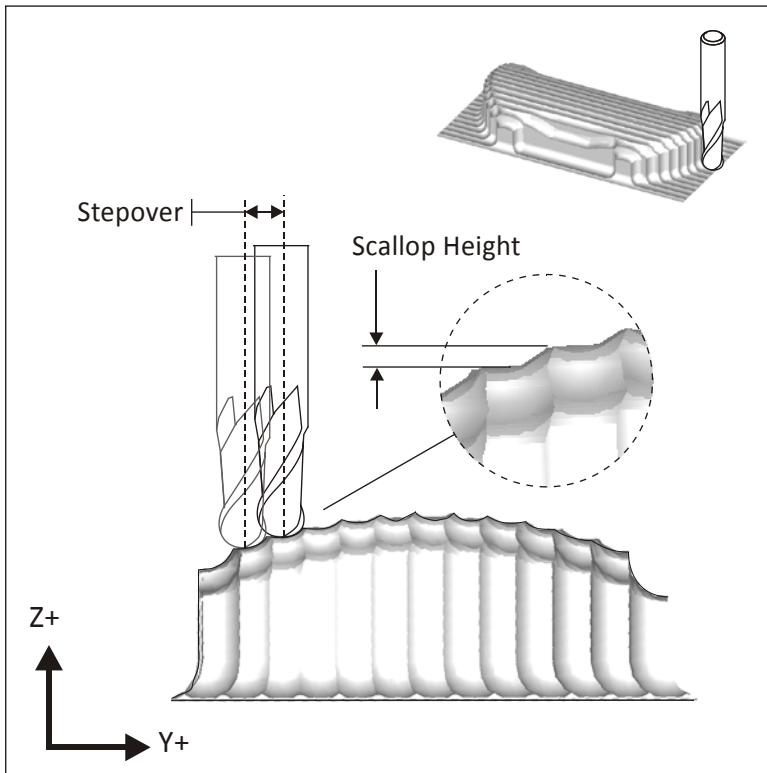


Figure 7: Scallop Height

9.6 – 3D Scallop

Scallop tool paths, when viewed from the top, appear similar to 2D spiral pocket tool paths. Scallop paths continually change the stepover distance to maintain a constant scallop height over the entire part.

Scallop tool paths are calculation intensive and may not work on all shapes, and produce very large CNC programs with many short moves.

Yet, when applied properly, they work very well and produce a superior surface finish.

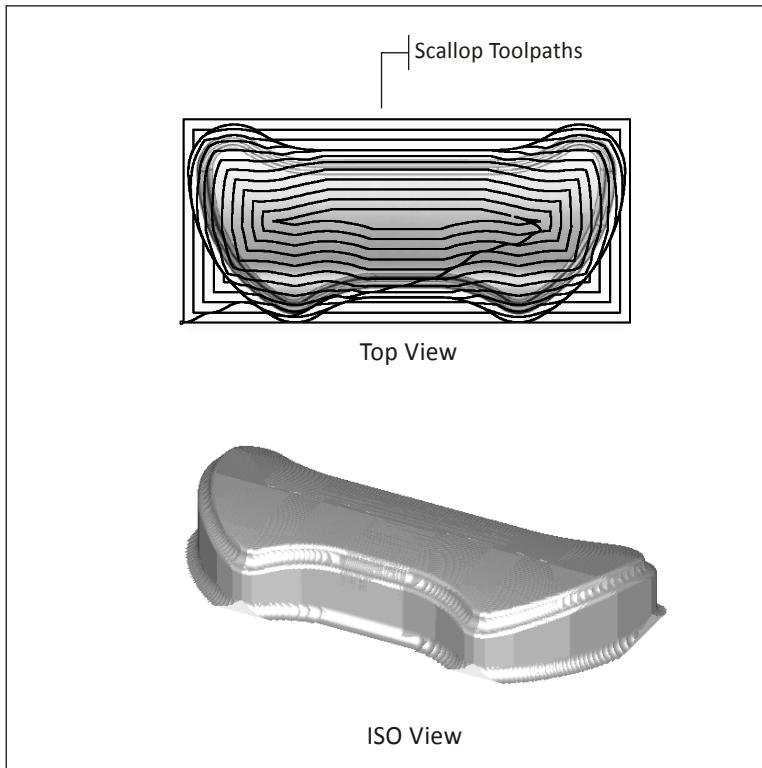


Figure 8: Scallop Tool Path

9.7 – REST Milling

Rest Milling is an acronym for **R**Emaining **S**TOck machining. Rest tool paths only remove material left by previous machining operations. They do this by calculating what stock has been previously removed and comparing it against the finished model. This is far more efficient than re-machining the entire part with a small tool just to create a few small features.

REST paths are calculation intensive. A good practice whenever possible is to use a tool whose diameter is slightly smaller than the smallest feature to be machined. This makes the REST calculations simpler and more effective.

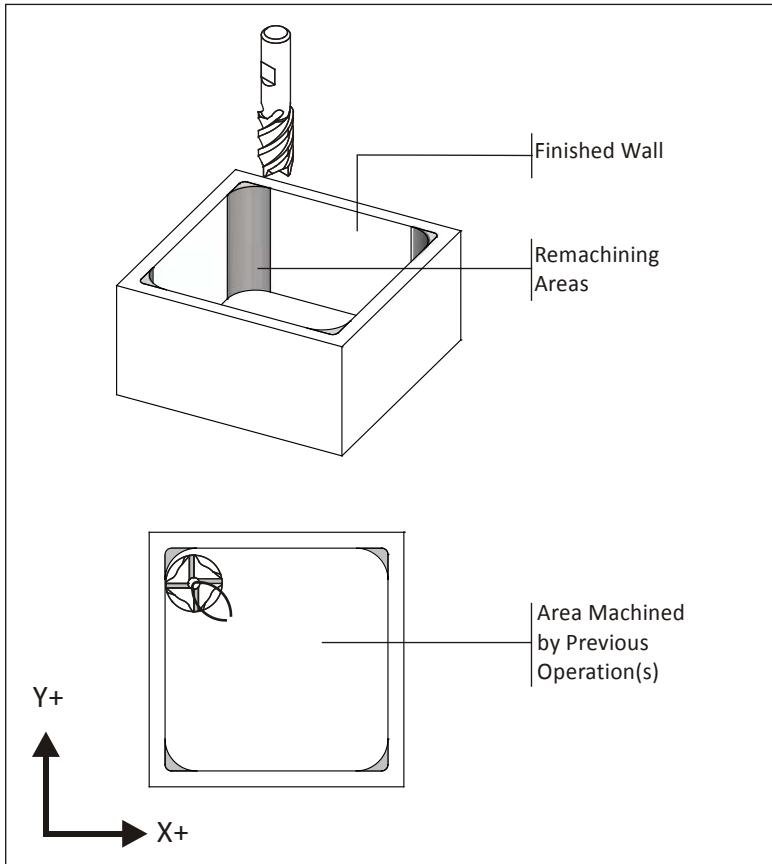


Figure 9: REST Milling

9.8 – Pencil Toolpaths

Notice in Figure 8 that Scallop tool paths may also leave areas that require further finishing, though not as much as, say, a Parallel path.

Pencil Trace tool paths are a type of remaining stock machining (Rest Milling) that works by tracing a tool along the seams between surfaces that form an inside angle. Figure 10 shows a pencil tool path between along the inside fillet. This clears any remaining scallops in this area, leaving a perfect seam.

Like with REST machining, use a tool smaller than the radius when possible for Pencil tool paths.

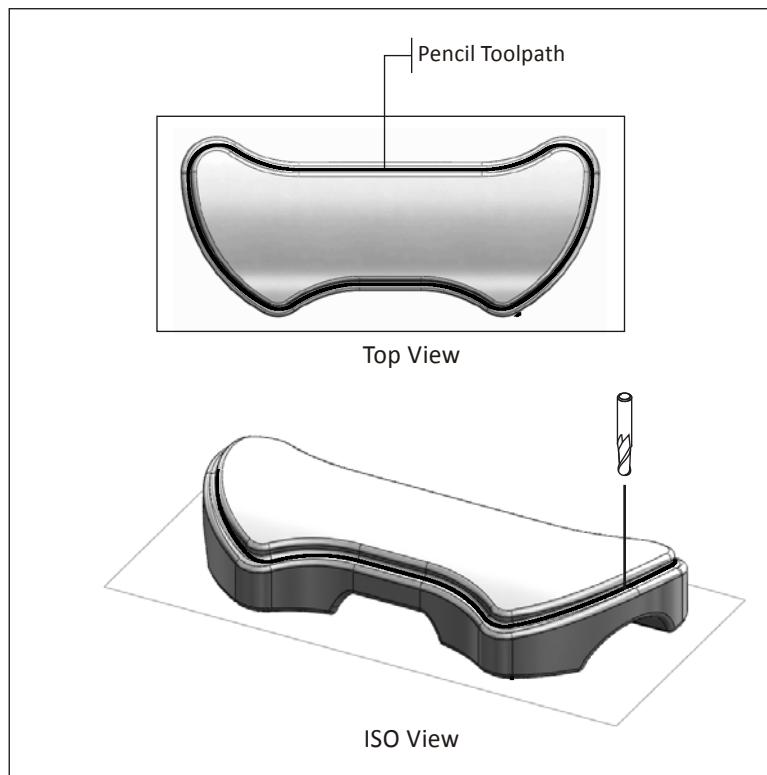


Figure 10: Pencil Trace Tool Path

9.9 – Conceptualizing 3D Toolpaths

There is a story about a man admiring an artist cutting the sculpture of a bear from a log. Impressed and amazed by the artist's ability, the man inquired how he was able to create such realism. The artist replied, "It's easy, I just cut away anything that does not look like a bear!"

3D machining is a similar mindset. You begin with a block of material and cut away anything that does not belong. Begin by getting rid of excess material as quickly and efficiently as possible. Try to leave a constant thickness of material for finish operations. Finish as much of the part as possible using the largest tools possible. Then finish machine finer features and details using Parallel or Scallop paths contained by 2D profiles, or REST and Pencil tool paths.

3D machining can be very challenging, but in many cases it is easier than many 2D parts. Some 2D parts involve scores of machining operations that require considerable forethought and work. Most 3D parts require fewer operations and these are largely automated by CAD/CAM software functions.

The key to success is planning ahead and preparing the model before creating tool paths. In the case of 3D machining, perhaps 50-80% of the time programming is actually CAD-related, getting the model ready for tool paths: creating runoff surfaces, suppressing features, creating check surfaces, and the like.

Don't begin making tool paths until you have a credible plan. If your planning is thorough and the CAD model is simplified so only those features to be machined are displayed, creating tool paths is far easier and outcomes better. To aid planning, use the forms in Appendix C, CNC Planning Forms.

Lesson 10

Milling Setups

Upon successful completion of this lesson, you will be able to:

- Explain Identify commonly used CNC work holding components.
- Identify the parts and accessories of a CNC vise.
- Explain the difference between vise hard jaws, step jaws and soft jaws.

1.1 - Overview

While there are many ways to hold a part during machining the ones illustrated in this chapter are the most commonly used for prototype and short-production machining. They work well for the types of parts you are likely to make.

Work holding for prototypes is often different than that for production machining. Large production lots allow the cost of tooling and fixtures to be amortized over many parts. While it is worth investing in complex fixtures to save seconds when making thousands of parts, it is not making only a few.

The goal with prototypes is to get the job done with minimal investments of time and money. This means using off-the-shelf components when possible and using methods that do not require a lot of time and effort.

There is often more than one work-holding method that will work for any given part. Review a tool supply catalog to familiarize yourself with the many types of clamps, bolts, and other holding devices that are available. The method you choose depends on many factors including personal preference, what work-holding components are readily available and cost.

In the end all that matters is that the method is safe, that it works, and that it is cost effective. Devising fixtures can be challenging. It requires being creative and thinking completely through all steps to machine the part. If you machine many parts, you will likely mix and match ideas learned here or from other sources.

1.2 Fixture Components

There are many fixture components for milling machines. This section covers some of the most common. If possible, acquire a fixture components catalog or look at the web sites of Carr-Lane (carrlane.com), Jergens (jergensinc.com), Chick (chickworkholding.com) MSC (mscdirect.com) and others.

Subplates and Angle Plates

A **Subplate** is a ground aluminum plate that bolts to the top of the machine table. It has threaded holes and bushings at regular intervals.

Subplates protect the machine table and speeds setups by allowing clamps and other components to be quickly and precisely located anywhere in the workspace.

Once installed, the subplate generally remains permanently fixed on the table. Fixtures and vises are installed on top of it.

An **Angle plate** is a precision ground steel plate that allows the part to be set on its side. Angle plates can point in a direction parallel to either the X or Y axis.

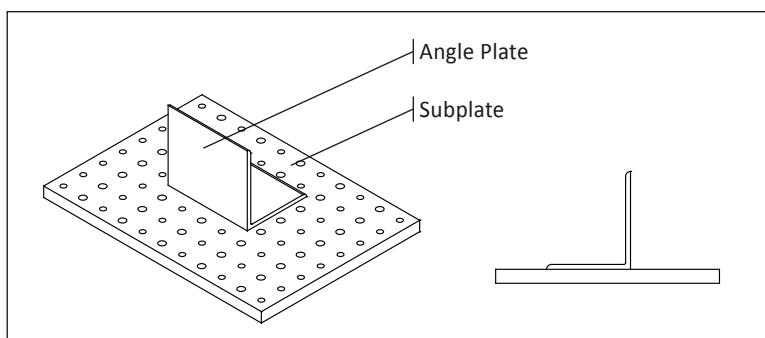


Figure 1: Angle Plate and Sub Plate

Clamps

There are literally thousands of types, styles, and sizes of **clamps** available. They are inexpensive, reusable, and versatile. You tool cabinet should include many types and sizes of clamps and accessories.

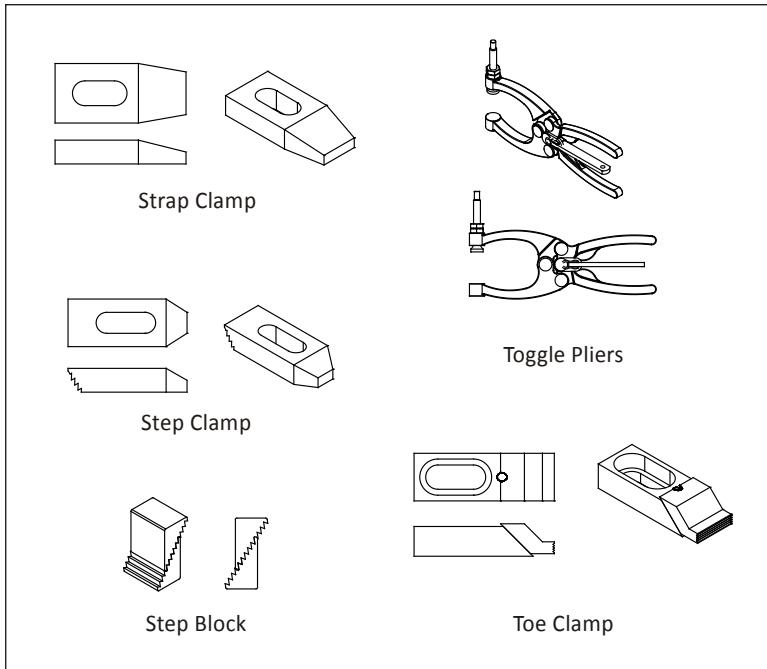


Figure 2: Clamps and Accessories

Strap Clamps exert downward pressure on the part. They are usually secured to the table by a special bolt that can be positioned anywhere along the T-slots in the table. One end of the clamp rests on the part, and the other on a step block.

To prevent galling an aluminum part, place a pad of aluminum between the clamp and part.

Step blocks have grooves that allow them to be stacked to different heights. Step blocks are used to support one end of the strap clamp.

Step Clamps are similar to Strap clamps, but include grooves that interlock with a single step block.

Toe Clamps bolt to the machine table. The nose of the clamp includes grooves to grip the part. It is extended by turning a screw, forcing the clamp against the part and downward.

Toggle Pliers are similar to hardware store vise grip pliers.

Shoulder Bolts and Dowel Pins

Dowel pins are used to precisely locate a part. They include a small chamfer to make insertion easier. Shoulder bolts both locate and grip the part. The ground shoulder on the bolt slides into a counterbore in the fixture.

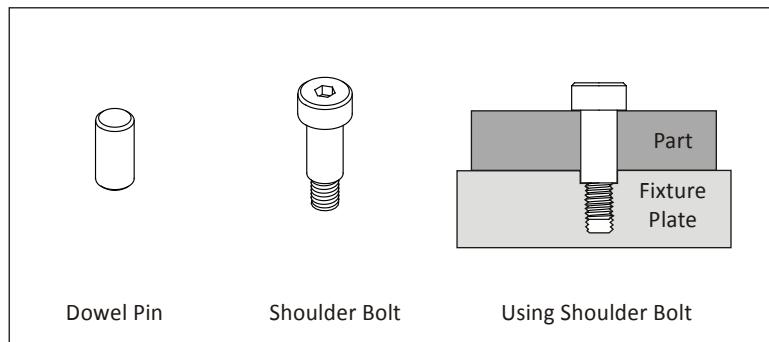


Figure 3: Dowel Pin and Shoulder Bolt

Vise and Accessories

The CNC vise is precision engineered and manufactured with components ground flat and perpendicular to within .0002 inches. The most common is referred to as a six inch vise, because the width of the jaws is six inches.

Once the vise is bolted to the table and aligned, parts are loaded into the vise and clamped by closing the jaws. The vise can exert tremendous force, so care is taken not to over-tighten the vise and deform fragile parts. Vise pressure must be appropriate to the part being held and expected cutting forces.

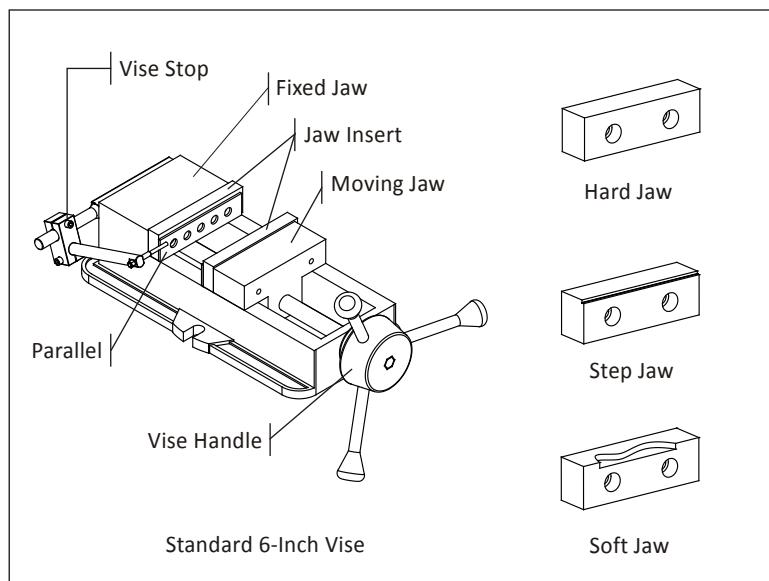


Figure 4: Vise and Accessories

The **Fixed Jaw** remains stationary. The **Moving Jaw** opens when the **Vise Handle** is turned. It is a good practice to remove the vise handle after the jaws are closed and before running the program. This is done by simply sliding the handle off.

A **Vise Stop** is a device that allows the parts to be loaded into the vise precisely. This image shows a style of vise stop that is particularly useful because it is adjustable up-down and left-right.

Hard Jaws are made of hardened steel and precision ground on all sides. They are usually used along with parallels.

Parallels are thin steel plates, available in various widths, used to set the grip length of the vise jaws.

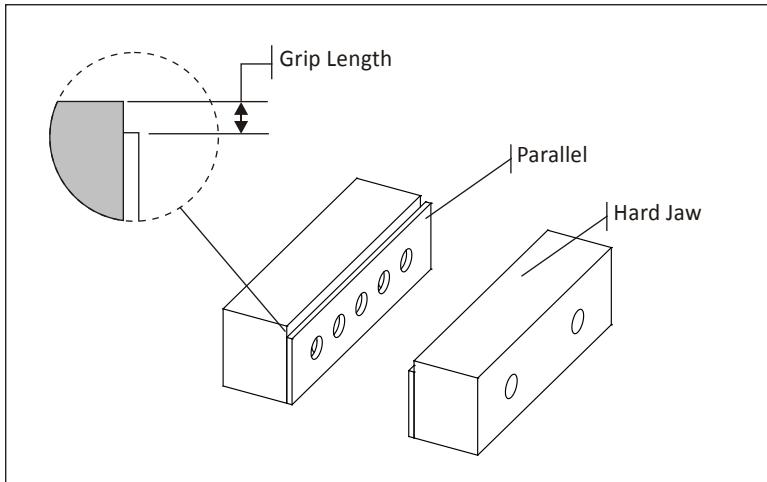


Figure 5: Hard Jaws

Step jaws are similar to hard jaws but include a step feature that eliminates the need for parallels.

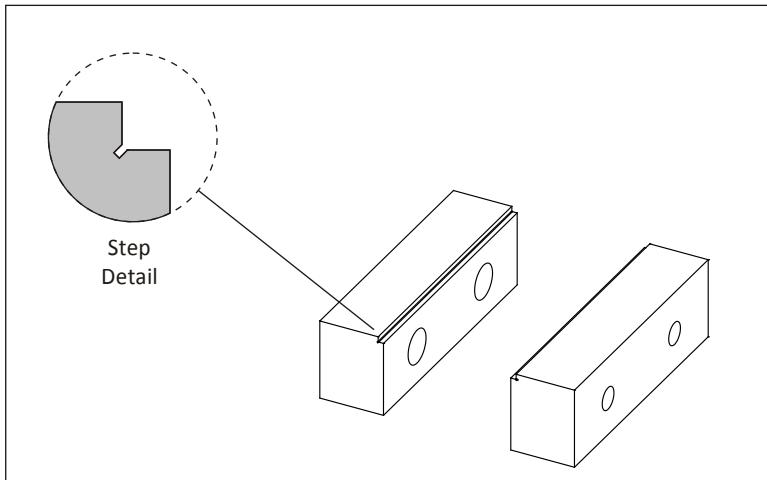


Figure 6: Step Jaws

Soft jaws are blanks of aluminum used to grip parts that cannot be held using hard jaws. A cutout the same shape as the part is machined into the soft jaws to grip irregular shapes.

When machining the cutout, place a bar between the jaws to set the correct spacing. Use a torque wrench or mark the vise so it can be closed with the exact same pressure each time a new part is loaded. Remove the spacer before clamping the part.

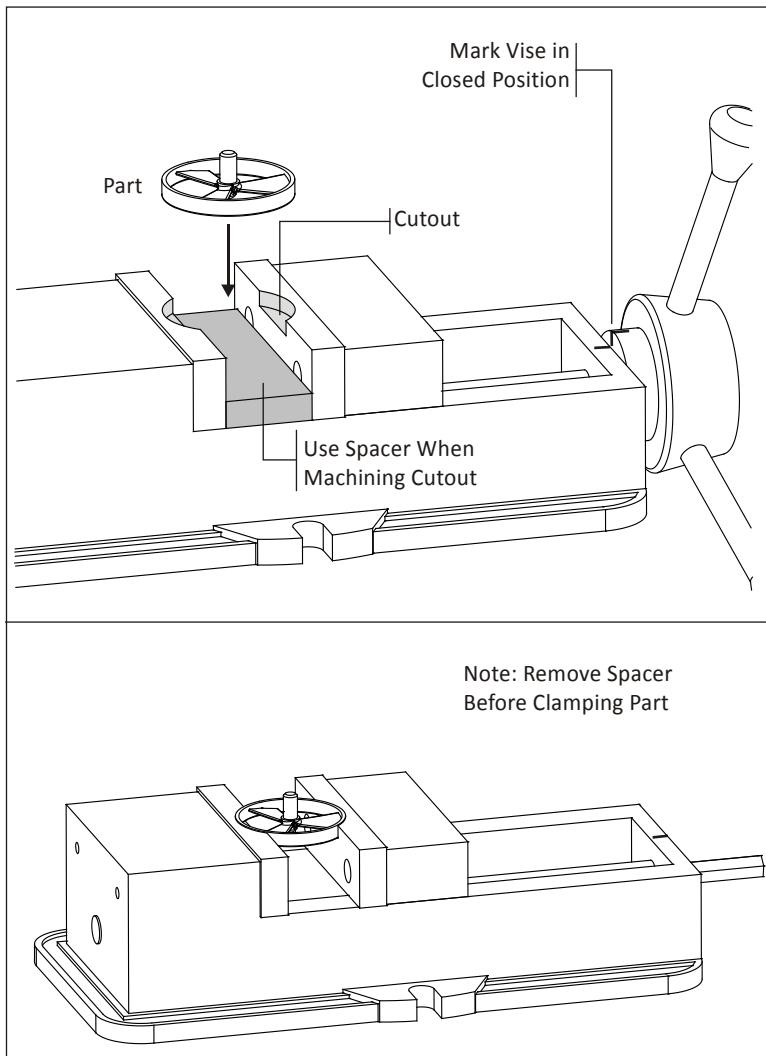


Figure 7: Machining Soft Jaws

Appendix A

CNC Milling Work-Holding Examples

Upon successful completion of this lesson, you will be able to:

- Explain how to properly align a CNC vise.
- Explain how to machine a block of stock material into a perfect rectangular prism.
- Machine a simple 2D contour applying cutter diameter compensation.
- List the types of vise jaws and their use.
- Install and align a CNC vise jaws.
- Use a vise to grip a part during machining.
- Make and use vise soft jaws to machine a 3D part.
- Machine multiple parts from a single piece of bar stock.
- Construct and use a fixture plate.
- Use clamps and bolts as a work-holding system.
- Machine a thin walled part.
- Set up and use an angle plate to machine a multi-sided part.
- Use shoulder bolts and dowel pins to machine a two-sided plate part.
- Use double-sided tape as a work-holding system.

Project 1: Install Vise

Before using a vise it must be aligned with the machine X-axis. This is a good practice before starting any new job to check vise alignment to ensure it is parallel to the machine axis.

1 Prepare Work Area

Jog the spindle out of the way and remove any tool to prevent accidentally bumping into it while working. Clean the machine table and bottom of vise to ensure there are no chips or debris between the table and vise.

2 Plan Vise Location

Locate the vise where all features on the part can be reached by the tool without exceeding the machine travel limits. Include overrun for lead in/out moves.

Before installing bolts, jog the machine to ensure the vise cannot contact any machine sheet metal throughout its travel.

3 Pre-Tighten Bolts

Insert bolts in T-slots.

Firmly tighten the left bolt. This bolt will act as a pivot for the vise to rotate about during alignment. Lightly tighten the right bolt.

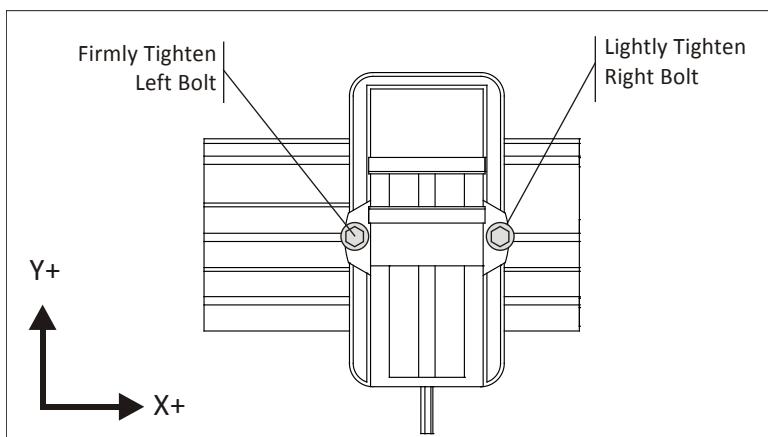


Figure 8: Pre Tighten Bolts

3 Align Vise With X-Axis

Install a dial indicator. Jog the machine until the dial stylus is along the fixed vise jaw. Jog the machine left to right along the vise jaw. Lightly tap the vise with a rubber mallet, causing it to rotate about the left bolt, until the indicator remains stable.

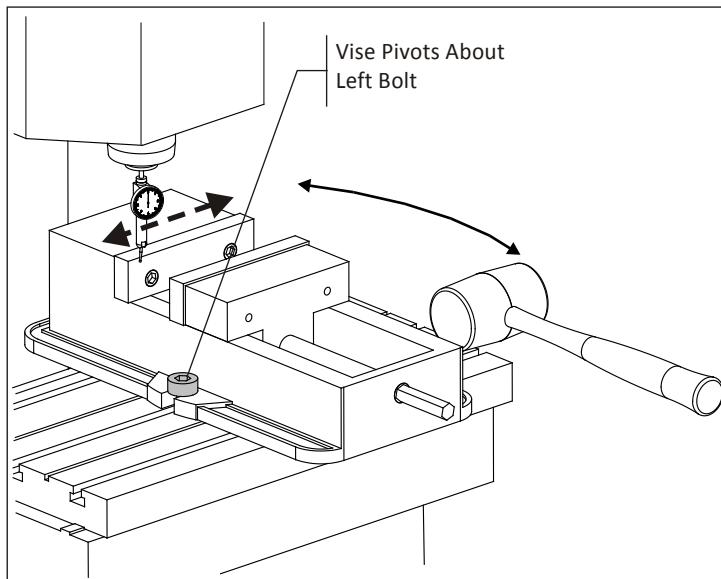


Figure 9: Align Vise

4 Tighten Bolts and Check Alignment

Tighten the right bolt fully, and then tighten the left bolt. Recheck the alignment by running the dial indicator along the fixed vise jaw and confirm the dial does not move more than .001 inches over the full width of the jaw.

Project 2: Install Vise Jaws

Once the vise is bolted and aligned with the machine, vise jaws are installed using cap screws. Follow these steps to install vise jaws:

1 Clean Vise and Jaws

Ensure there are no chips on any ground surfaces of the vise or the vise jaws.

2 Install Bolts

Install Cap screws to secure the vise and Hand-tighten both bolts.

3 Tap Down Jaws

Use a rubber mallet to tap down the vise jaws so there are no air gaps between the jaws and the vise face.

4 Fully Tighten Bolts In Correct Order

Tighten the bolts in the order shown, ensuring the jaws do not lift as the bolts are firmly tightened.

5 Check Work

Any slippage of the jaws is not acceptable. Ensure all bolts are very tight before using the vise. Try sliding a piece of paper or feeler gage between vise jaws and vise to ensure there is no space due to the jaw lifting.

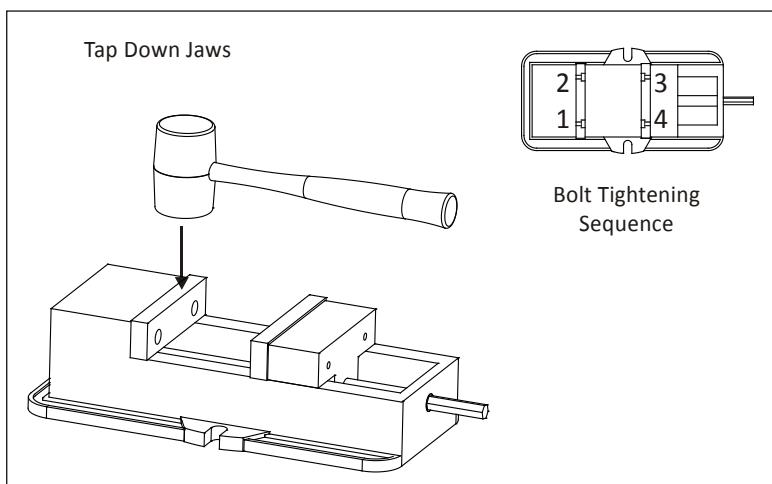


Figure 10: Installing Vise Jaws

Project 3: Square Block

For some prismatic shapes it is preferable begin by machining a block of material to a specific size. This can be done by jogging the machine using the hand wheel on the control. The process of squaring a block is also helpful for understanding concepts of flat, square and parallel as they relate to the CNC machine axes.

1 Prepare part for machining

Begin with sawed raw material large enough to be machined on all sides: usually at least .050 inches oversize in each dimension.

Stock walls must be sufficiently flat and opposite sides parallel to be safely gripped by the vise. Use common sense; if the part can be moved by hand in the vise, it will loosen under cutting pressure and could move or be ejected from the vise.

Faces are machined in the sequence shown in Figure 11 below. Figures on the following pages indicate the Top face in color grey, making it easier to see how the part is flipped for each operation.

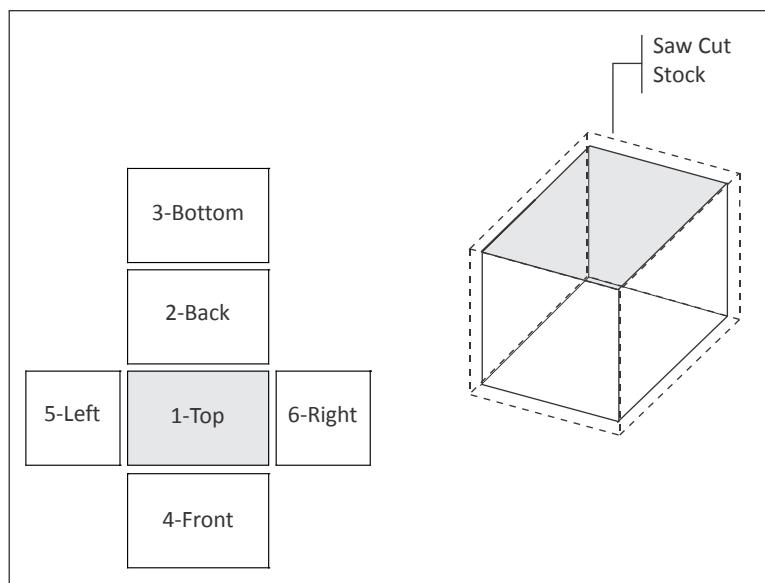


Figure 11: Saw Cut Stock and Machining Sequence

2 Set Tool Position

The machine spindle must be off. Manually jog the tool so it is about one inch above the Top face. Move the machine down in Z in .1 inch increments. After each move carefully slide a 1-2-3 block between the top surface and the tool, until the block makes contact.

Switch the jog increment to .001 and raise the tool until the block just slides underneath. This sets the tool exactly 1.00 inch above the face of the stock.

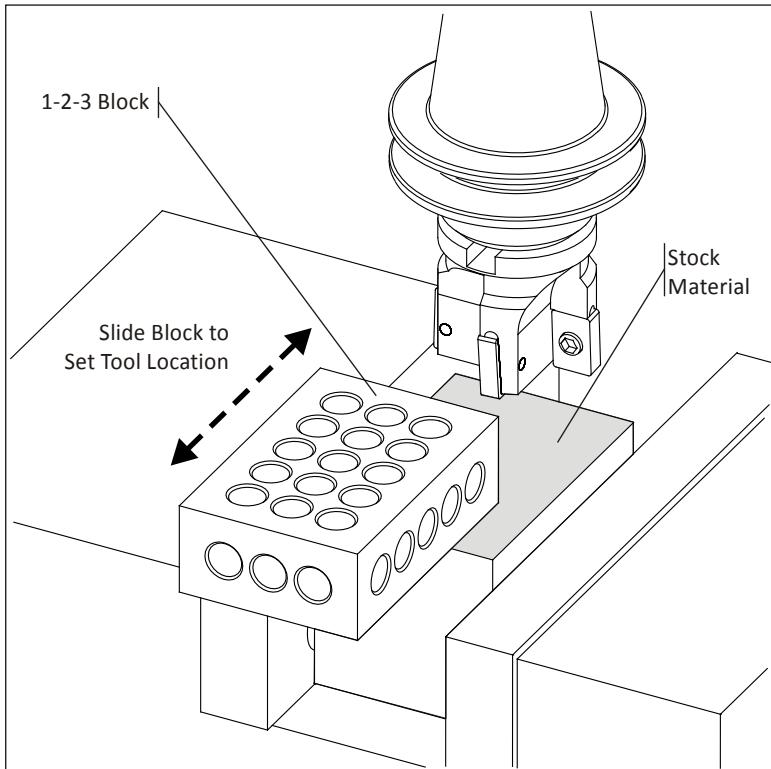


Figure 12: Set Tool Position

3 Machine Top Surface

Estimate the amount of material to remove with the first cut, and then jog the machine down 1.00 inches plus this value. For instance, to remove about .010 inches of material, jog the machine down 1.010 inches.

Set the jog control to the X-axis, turn on the spindle CW, and use the hand control to machine the top face. When complete, turn the spindle off and move the tool clear of the work area.

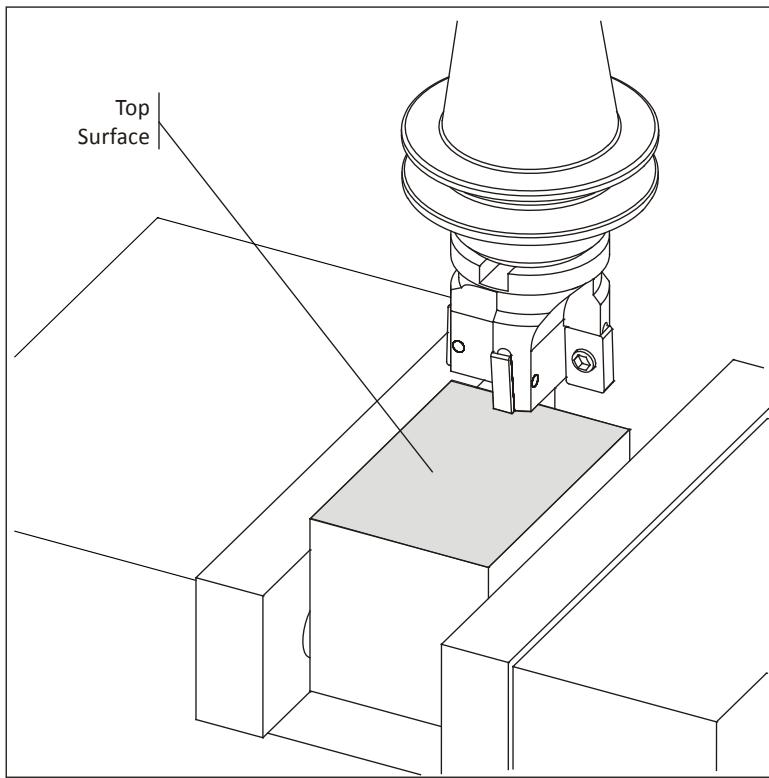


Figure 13: Machine Top Surface

4 Machine Front Surface

Flip the part about the X-axis as shown in Figure 14 below so the top surface machined in the previous step rests against the fixed vise jaw. Be sure to first clean the work area with a paintbrush to ensure the part does not rest on any chips or debris.

Because the surface against the movable jaw is rough, insert a solid brass rod as shown below. This forms a fulcrum that lets the part pivot, ensuring the top surface rests flat against the back fixed vise jaw.

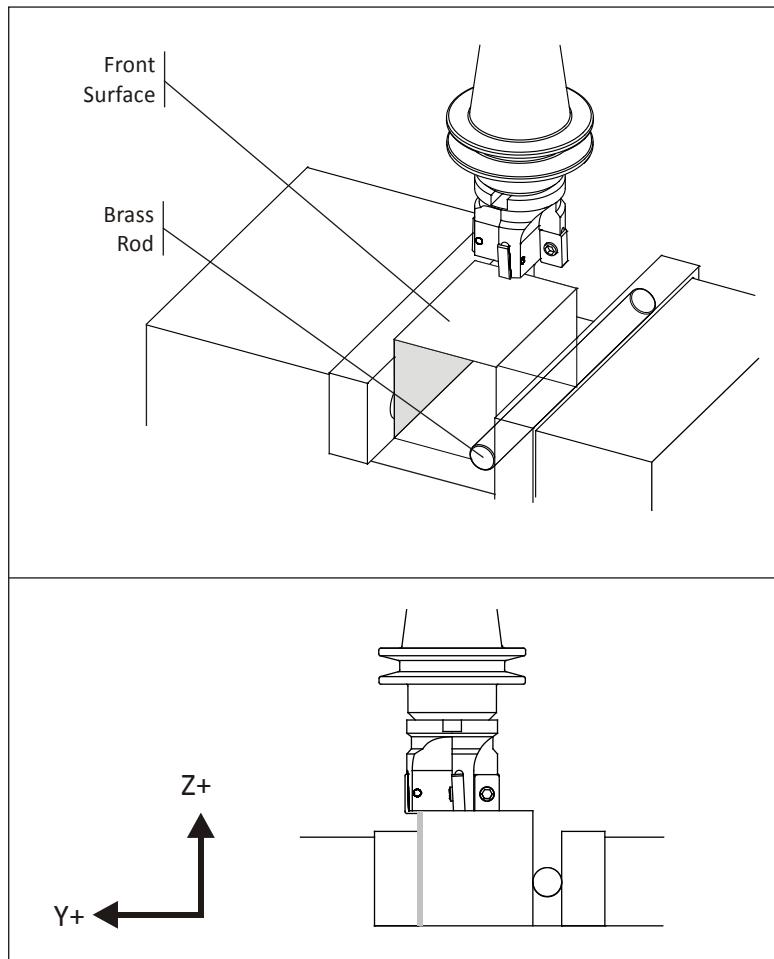


Figure 14: Machine Front Surface

5 Machine Bottom Surface

Flip the part about the X-axis as shown in Figure 15 placing the top surface against the floor of the vise or parallels. Again use the brass rod to ensure part lays flat against the fixed jaw, clear chips, and ensure there is no small gap between the top face and vise floor.

Rough and finish the bottom side as required to establish the finish dimension between the top and bottom faces.

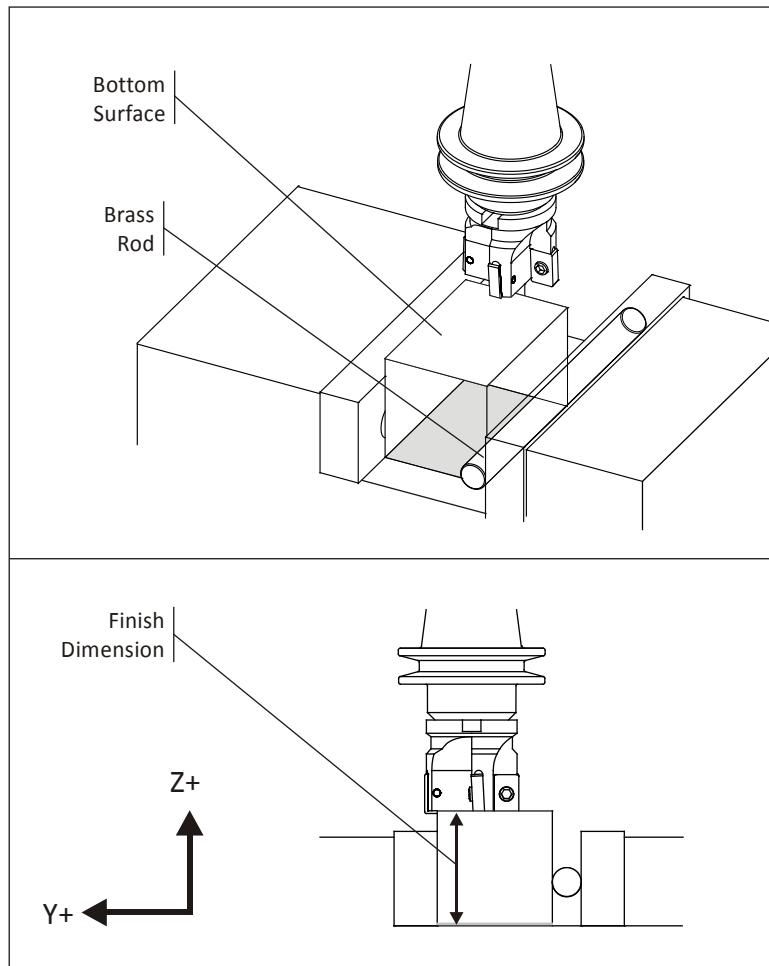


Figure 15: Machine Bottom Surface

6 Machine Back Surface

Flip the part about the X-axis as shown in Figure 16 so the top surface now rests against the moving jaw. Do not use the brass rod. Clear all chips and tap down the part.

Rough and finish this side as required to establish the finish dimension between the front and back faces.

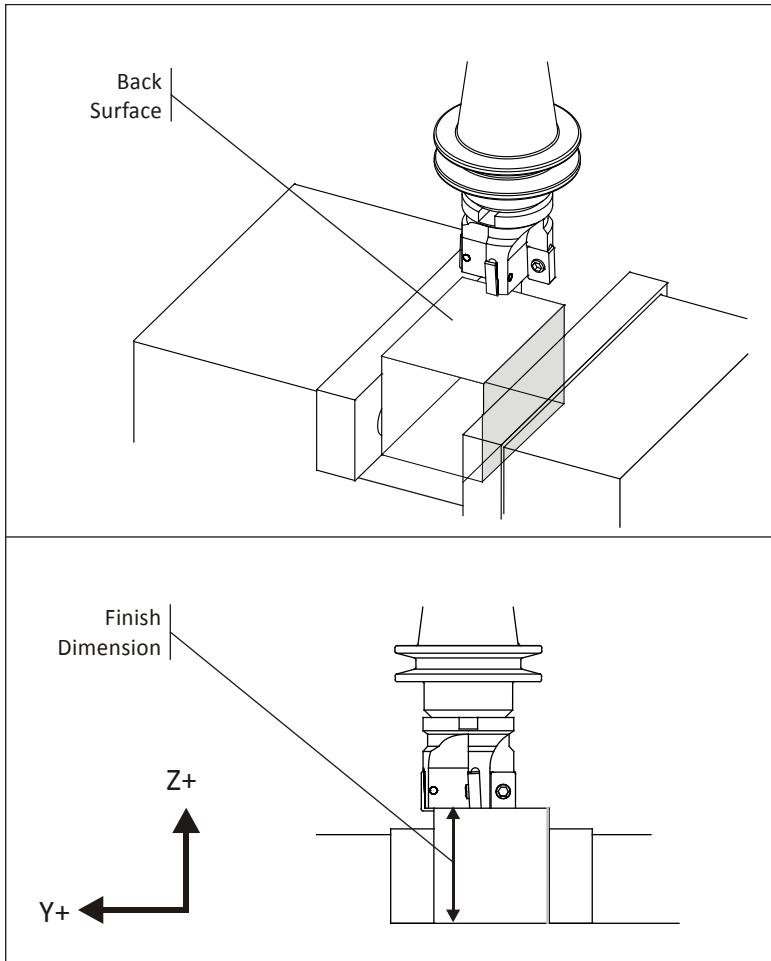


Figure 16: Machine Back Surface

7 Machine Left Surface

Next, rotate the part counterclockwise about the machine Y-axis standing it on end. Use a machinist's square to set the machined surface perpendicular to the vise base.

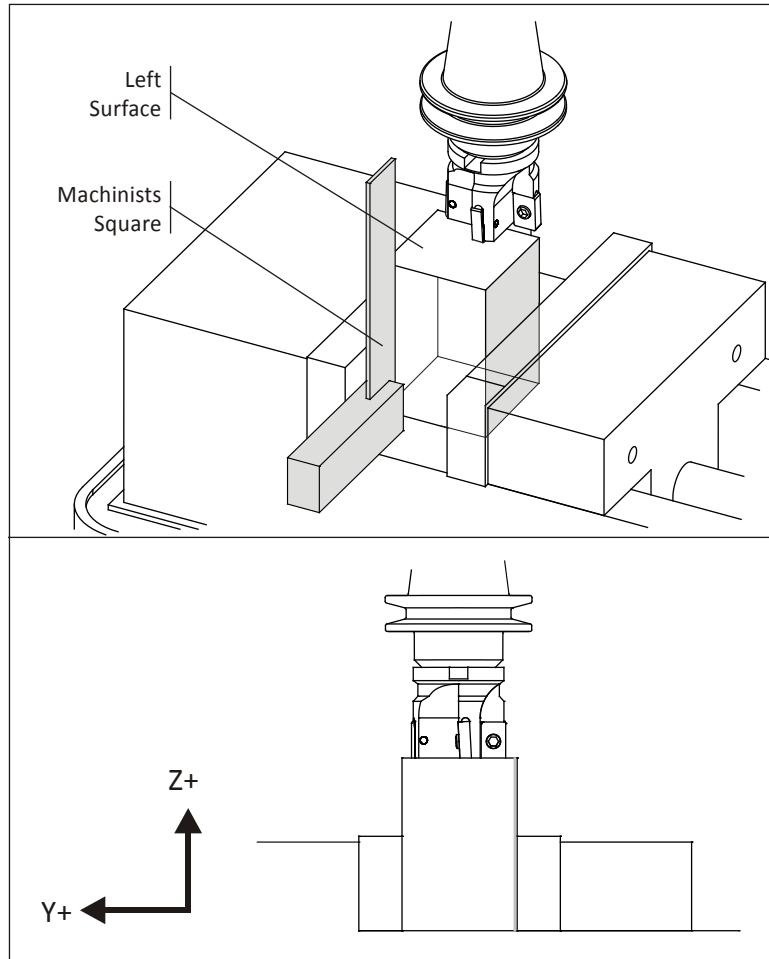


Figure 17: Machine Left Surface

8 Machine Right Surface

Finally, rotate the part again about the machine Y-axis 180 degrees so that the left surface is now against the machine base. Clear chips and tap down the part. Machine the right face to establish the finish dimension between the left and right part faces.

The part is now the precise size and all sides are flat, square, and parallel.

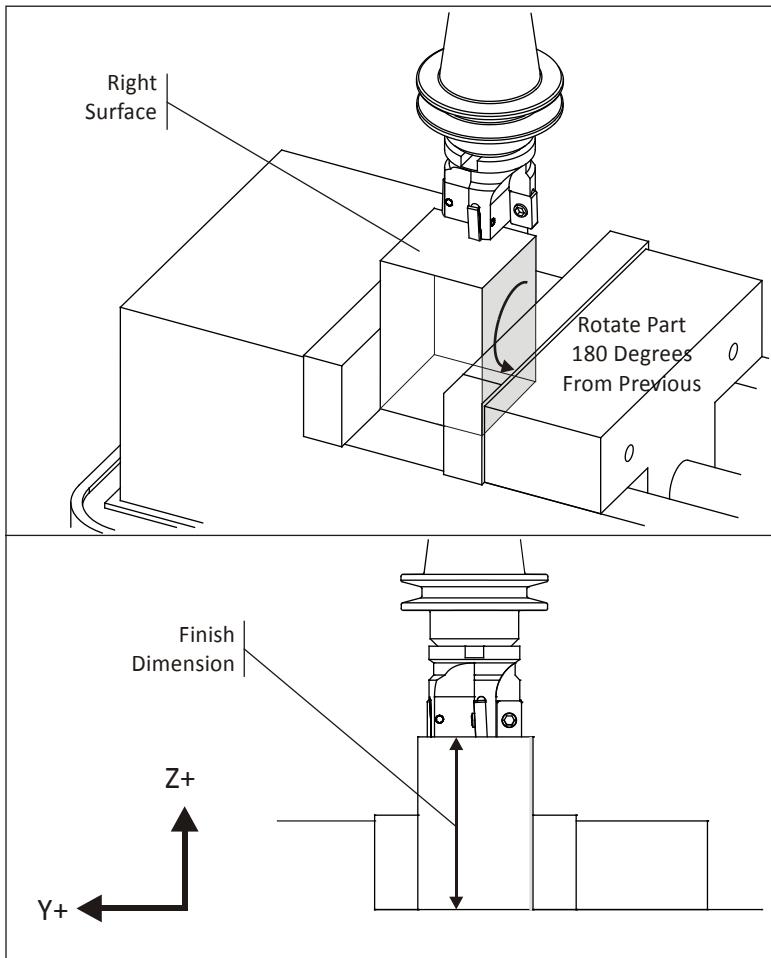


Figure 18: Machine Right Surface

Project 4: Contour Square Step

This exercise teaches how to do the following:

- Set the fixture offset.
- Set tool length offset (TLO).
- Adjust cutter diameter compensation (CDC).

The fixture offset is a point on the part that can be easily and accurately found by mechanical means. For example, in Figure 19, the upper-left corner of the stock can be located using an edge finder. The fixture offset (G54) is located at this point. If it were not, the datum could be shifted from this reference point by changing the G54 XYZ-values on the machine control.

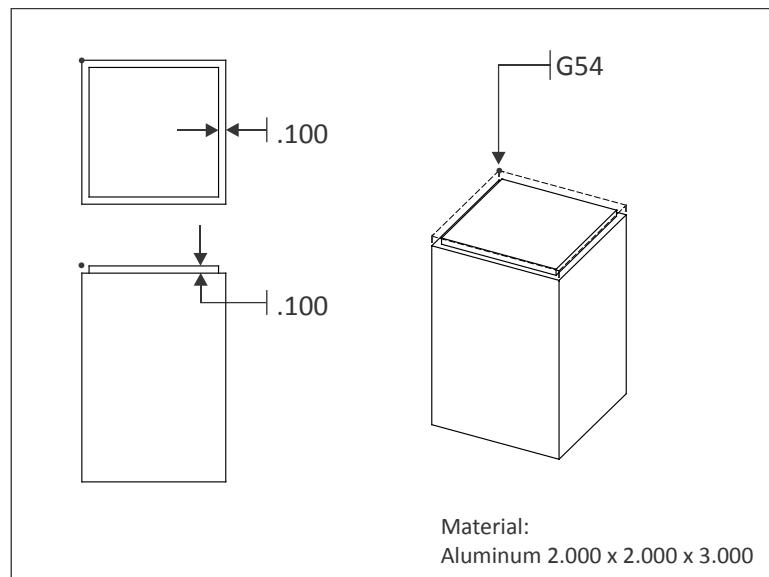


Figure 19: Contour Block

It is common to choose the upper-left corner of the stock as the fixture offset for the first operation. If the stock is sawed, ensure there is sufficient stock allowance around the part so finish operations remove material all around.

For subsequent operations the fixture offset is set from features machined in previous operations.

1 Square Block

Begin with a finish machined 2x2x3in block (created in Project 3).

2 Find Fixture Offset XY

Use an edge finder to locate the upper-left corner of the block (shown by the black dot in Figure 20).

3 Set Fixture Offset A

Set the fixture offset Z (G54) to the top of the block.

4 Program Contour Tool Path

Create a contour tool path to mill a .100x.100 step around the part as shown above. Use Wear compensation, line/arc lead in/out moves, and the correct cutting speed and feed for the tool. Use a finish pass of XY .010 and Z.005 to ensure the wall and floor are dimensionally accurate and have a good surface finish.

5 Adjust CDC

After machining the contour, measure it with a dial indicator. The step should be exactly $.100\text{in}$ $+/- .001$ in depth. If not, the G54 Z position or Tool Length Offset (TLO) was not set properly.

Check that the boss measures $1.800 \times 1.800\text{ in}$ $+/- .001$. If it is too large, it is likely because the tool is worn and thus not exactly $.250$ diameter. Adjust for this wear by changing the wear compensation (CDC) for this tool on the control, and then re-running the program.

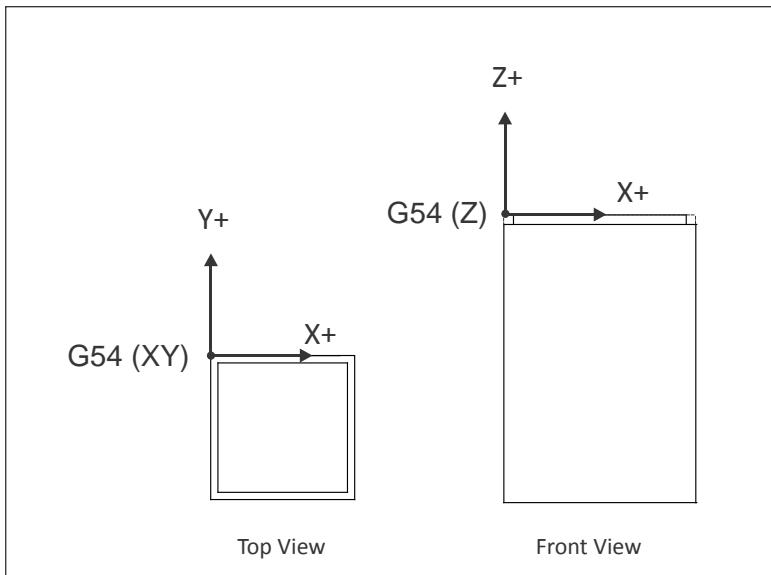


Figure 20: Set G54

Contour Block: Job-1 Setup

G54

G54 Datum: Upper-Left corner of previously finish machined $2 \times 2 \times 3\text{in}$ block.

Op-1 Contour	Tool (in) .25 End Mill 4-Flute	Speed (rpm) 7500	Feed XY (ipm) 60.	Feed Z (ipm) 30.
Rough and finish the OD contour using wear compensation. Make one roughing pass and then a finish pass that removes $.010\text{in}$ stock on walls, and $.005\text{in}$ on floor.				

Figure 21: Contour Tool Path

Projects 5-7 Overview

Projects 5, 6 and 7 form a simple assembly. The parts must be machined to the specified tolerances in order to fit together.

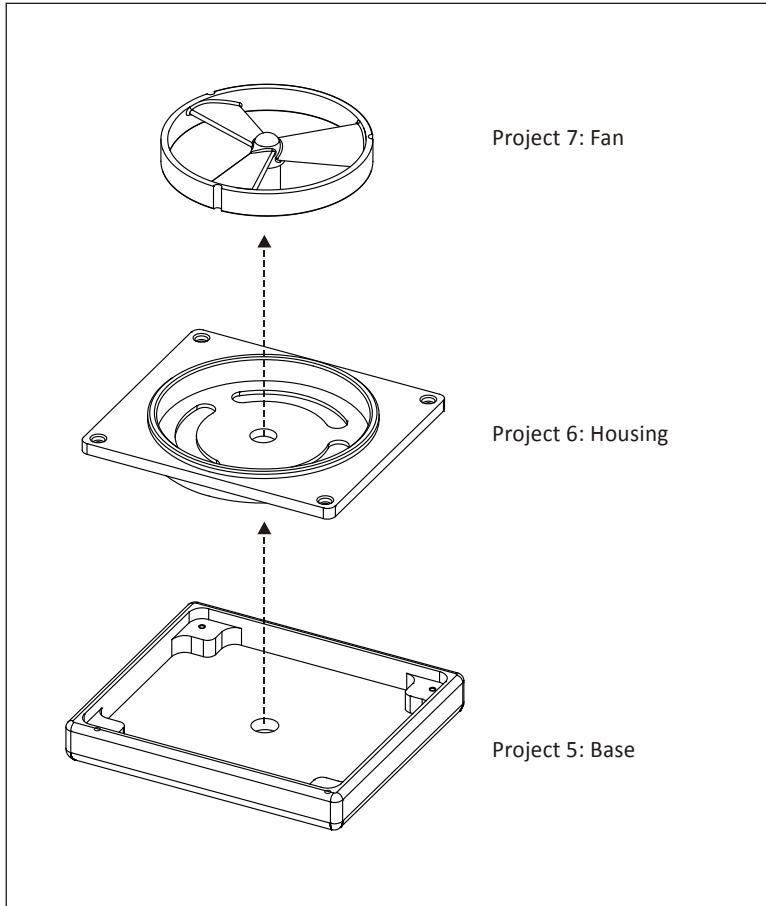


Figure 22: Fan Assembly, Projects 5-7

Project 5: Base

This project teaches the following skills:

- Face milling.
- Drilling.
- Contour.
- Pocket.
- Flip part in vise and pick up second datum.

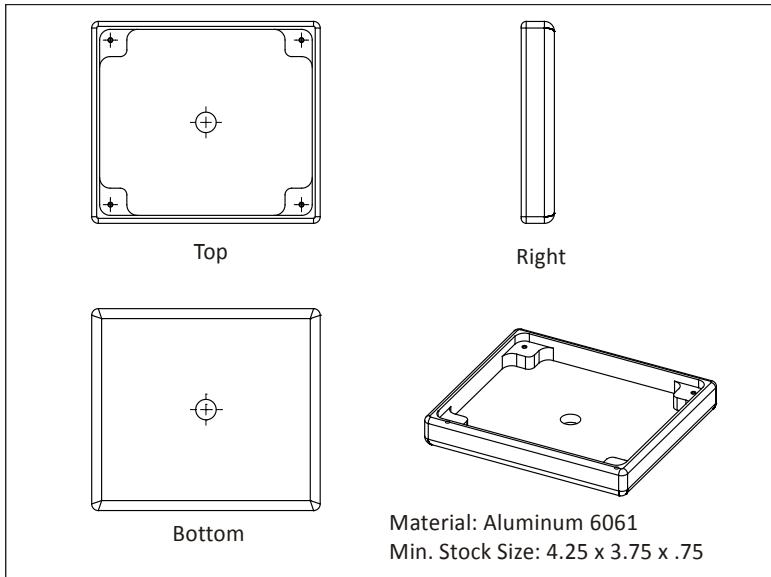


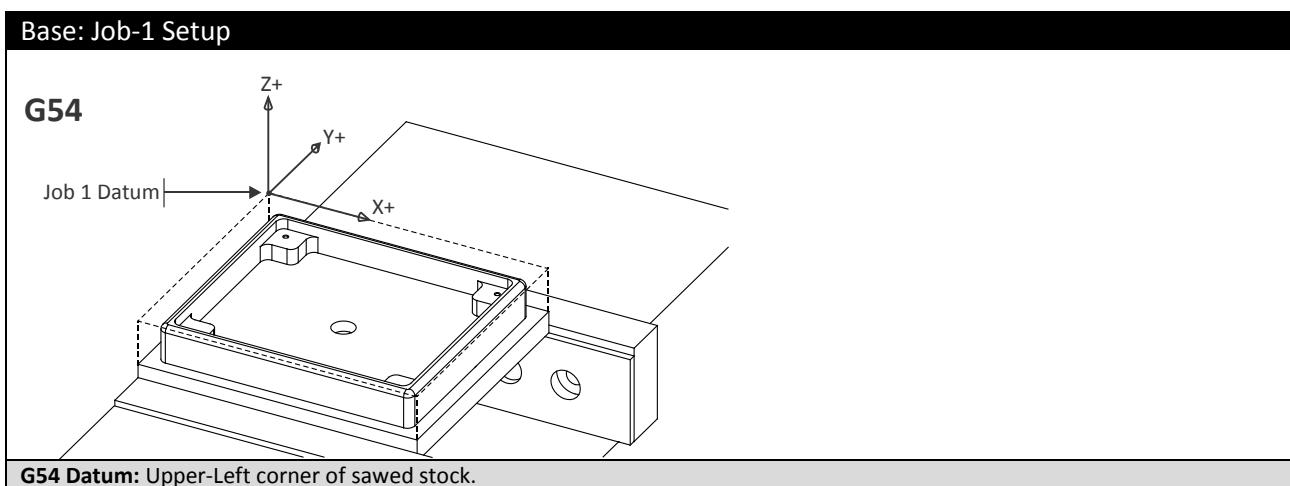
Figure 23: Project-5, Base

1 Job 1 Setup

Grip the sawed stock as shown in Figure 24. Set the XY-datum from the upper-left corner of the stock. Make sure there is sufficient stock so that finish tool paths remove material all around the part (typically .01-.05 inches from the stock corner). Set the Z-datum so that the facing operation removes material from the face of the part (typically .01 below the face of the sawed stock).

2 Job 1 Operations

Machine the part using the processes and parameters in Figure 24. If needed, select tools, speeds and feeds compatible with your machine.



Op-1 Face	Tool (in) .375 End Mill 2-Flute	Speed (rpm) 8150	Feed XY (ipm) 33.	Feed Z (ipm) 15.
Use a stubby end mill to reduce chatter (flute length=.625). Stepover=.20, Take a Z finish pass of .005 to ensure a good surface finish.				
Op-2 2D Contour	Tool (in) .375 End Mill 2-Flute	Speed (rpm) 8150	Feed XY (ipm) 33.	Feed Z (ipm) 15.
Contour OD. Rough Stepover =.2, Stepdown=.2. Finish XY=.01, No Finish Z needed. Number of rough passes depends on stock size.				
Op-3 2D Pocket	Tool (in) .375 End Mill 2-Flute	Speed (rpm) 8150	Feed XY (ipm) 33.	Feed Z (ipm) 15.
Rough and finish pocket with islands. Rough Stepover =.2, Stepdown=.2. Finish XY=.01, Finish Z=.005				
Op-4 2D Contour	Tool (in) .375 End Mill 2-Flute	Speed (rpm) 8150	Feed XY (ipm) 33.	Feed Z (ipm) 10.
Contour the hole in the center of the pocket with helical entry. Finish XY=.01, Finish Z -.010 below bottom so no flashing is left after facing opposite side.				
Op-5 2D Contour	Tool (in) .125 Center Drill, 90Deg	Speed (rpm) 5000	Feed XY (ipm) 40.	Feed Z (ipm) 20.
De-burr top edges by creating a .07 chamfer. Use line/arc lead in/out.				
Op-6 CTR Drill	Tool (in) .125 Center Drill, 90Deg	Speed (rpm) 6100	Feed XY (ipm) N/A	Feed Z (ipm) 12.
Center drill the four #2-56 holes. Dwell .5 seconds.				
Op-7 Peck Drill	Tool (in) Drill .07 Dia. (#50)	Speed (rpm) 10000	Peck Inc.(in) .035	Feed Z (ipm) 20.
Peck drill four #2-56 holes.				
Op-8 Rigid Tap	Tool (inch) #2-56 Tap	Speed (rpm) 4300	Peck Inc. N/A	Feed Z (ipm) 76.786
Tap four holes to depth Z-.26 below top of boss. If your machine does not support rigid tapping, tap by hand after job is complete.				
Op-9 Contour	Tool (in) .115x.093 Radius Mill	Speed (rpm) 6640	Feed XY (ipm) 40.	Feed Z (ipm) 20.
Use corner round tool to create a fillet on outside of the part. Take two finish passes (one spring pass) to produce a very good surface finish.				

Figure 24: Base, Top Side Machining Parameters

3 Job 2 Find XY Datum

The XY datum for Job 2 could be set at the center of the through hole using a coaxial indicator, dial indicator, or machine part probe. The method illustrated here shows how to set the datum using a vise stop and 1-2-3 block.

Program X0 is found using an edge finder to locate the left edge of the 1-2-3 block (YZ Data Reference Frame). Set the Fixture Offset X-value as the centerline of the left 1-2-3 block edge.

Program Y0 is found using an edge finder to locate the fixed vise jaw (XZ Data Reference Frame). Set the Fixture Offset Y-value as the centerline back 1-2-3 block edge.

Step-by-step instructions for setting the Fixture Offset XY using an edge finder are found in Chapter 6, CNC Operation: Set Fixture Offset XY.

Many CNC machines are now equipped with a Part Probe. This instrument is gripped in machine spindle and communicates with CNC control when the probe stylus touches against a surface.

Part probes greatly speed and simplify setting fixture offsets. To learn more refer to the documentation included with your part probe or search YouTube© for videos on the topic.

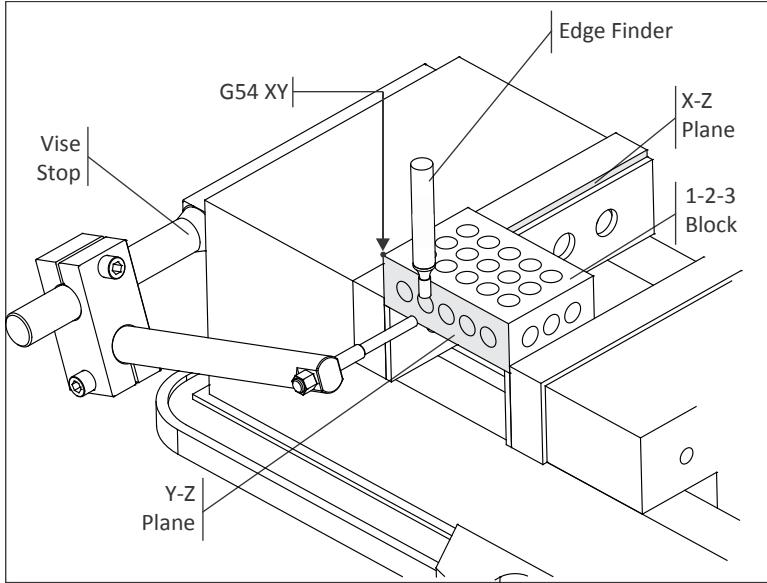


Figure 25: Set G54 XY Using A 1-2-3 Block

4 Find Program Z Datum

G54 Z could be set at the top of the vise parallels. If using this method, set the tool rapid and clearance heights sufficient to avoid colliding with the part, vise, and fixture.

The approach described here sets G54 Z to the value of the face of the finished part for Job 2. To find the distance from the top of a 1-2-3 block to the top of the finished part use this formula:

$$\text{Fixture Offset Z} = (\mathbf{A} - \mathbf{1.0} + \mathbf{B})$$

As illustrated in Figure 25, **A** is the incremental distance from the top of a 1-2-3 block resting on the table to the top of a 1-2-3 block set in the vise. **B** is the finished thickness of the part. The value **1.0** is the thickness of the 1-2-3 block.

Stated as a word problem, this formula reads, "The Fixture Offset Z is the distance from tool setting point to the bottom of the 1-2-3 block, plus the thickness of the finished part."

The process of setting the fixture offset is described in detail in Lesson 6, CNC Operation.

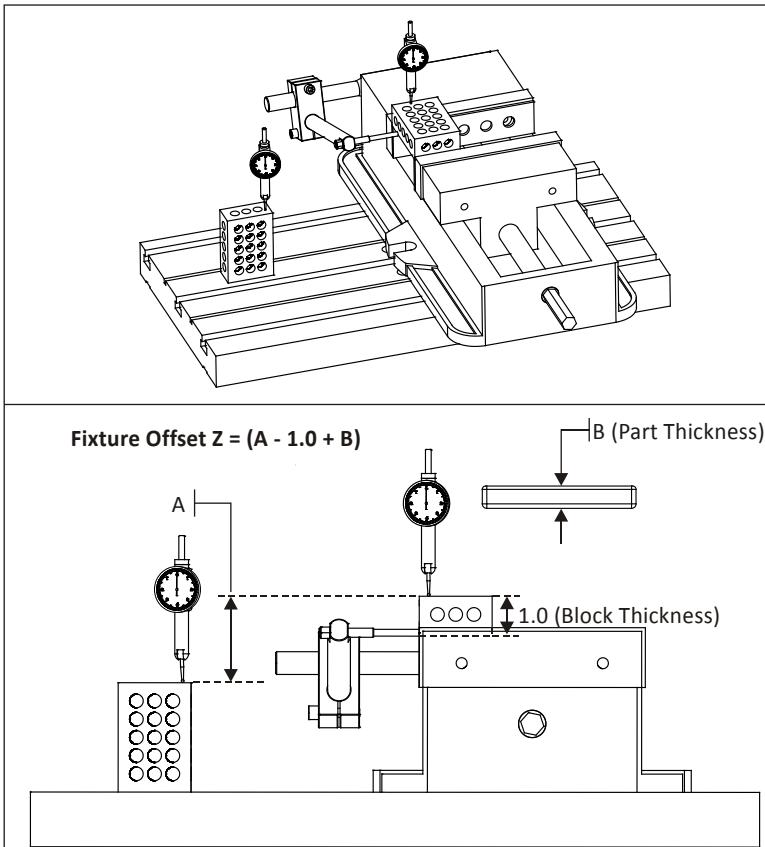


Figure 26: Set Z Datum

5 Job 2 Operations

Clean the work area and set the part finished upside-down in the vise. Slide the part left-right and apply pressure so it rests firmly against the fixed vise jaw and vise stop as shown in Figure 27 and close the vise jaws firmly. This establishes the XY datum. Next, tap the part down with a rubber mallet to ensure it is flat against the parallels and close the jaws tightly.

When making multiple parts, mark the vise with marker or use a torque wrench to ensure each part is gripped with the same vise force.

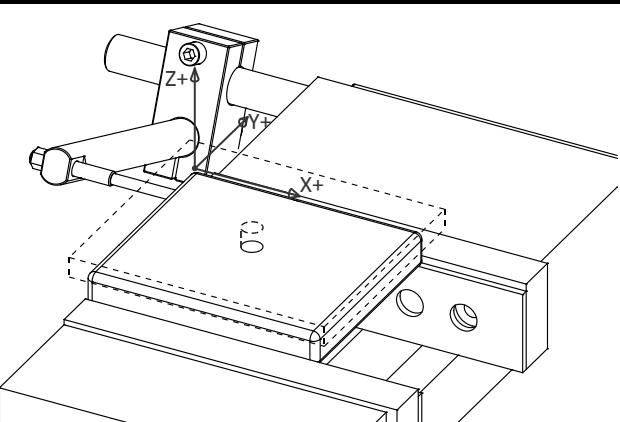
Base: Job-2 Setup				
G55				
G54 Datum: Upper-left corner of finished part face.				
Contour OD leaving .010in XY Stock. This cut is used to clear excess material from the part so the corner round tool will not engage an excessive amount of material.				
Op-1 2D Contour	Tool (inch) .375 End Mill 2-Flute	Speed (rpm) 9800	Feed XY (ipm) 68.	Feed Z (ipm) 30.
Face part using a stepover = .20in and stepdown = .10in . The number of Z-roughing passes required depends on the amount of material remaining on part. Finish pass should be no greater than .005in to yield a good finish.				
Op-2 Face	Tool (inch) .375 End Mill 2-Flute	Speed (rpm) 9800	Feed XY (ipm) 68.	Feed Z (ipm) 30.
Use corner round tool to create fillet on outside of part. Take two finish passes to produce a very good surface finish.				
Op-3 2D Contour	Tool (inch) .115x.093 Radius Mill	Speed (rpm) 5000	Feed XY (ipm) 40.	Feed Z (ipm) 20.
Engrave lettering. Depth=Z-.005in				

Figure 27: Base, Bottom Side Machining Parameters

Project 6: Housing

This project teaches the following skills:

- Multiple setups.
- 2D Remaining Stock (REST) machining.
- Slot milling.

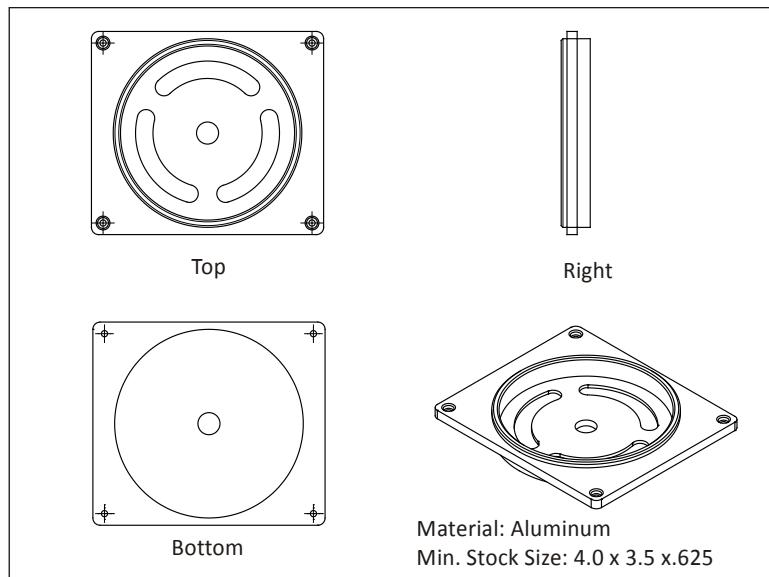


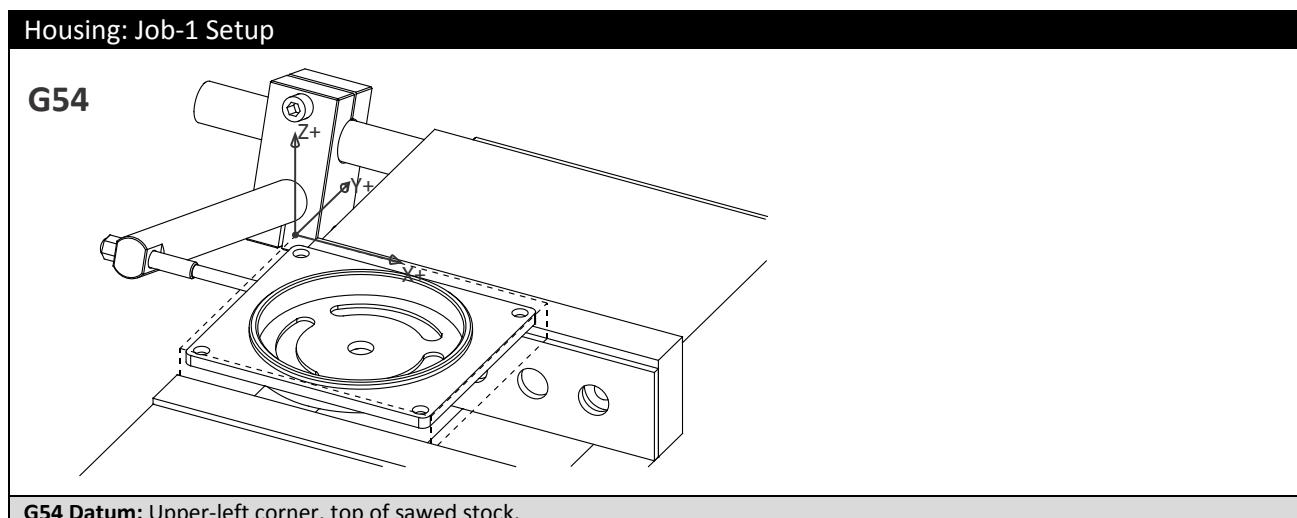
Figure 28: Project-6, Housing

1 Job 1 Setup

Grip the sawed stock in the vise as shown in Figure 29. Set the XY-datum off the upper left corner of the stock. Leave enough stock so the contour tool path removes material all around the part (typically .20-.05 inches from the sawed stock corner). Set the Z-datum so that the face tool removes material off the entire face of the part (typically .01 inches below the face of the sawed stock).

2 Job 1 Operations

Machine the part using the processes and sequence in Figure 29. Select appropriate tools, speeds, feeds and machining parameters.



Op-1 Face	Face part.
Op-2 Contour OD	Rough and finish OD of part (rectangle shape). Be sure to machine below the bottom of the OD so no flashing will remain after Job 2, but don't collide with the vise hard jaws.
Op-3 Contour OD	Rough and finish OD of round boss.
Op-4 Pocket	Rough and finish pocket to top of slots.
Op-5 Slot Mill	Rough and finish slots.
Op-6 Pocket	Rough and finish hole through center. Be sure to break through bottom by .01. This hole must be completely through the stock because it will be used to set the G54 XY for Job 2.
Op-7 Spot Drill	Spot and chamfer four holes.
Op-8 Drill	Drill clearance holes for #2-56 socket head cap screws.
Op-9 Counterbore	Counterbore with 3/16 (.188) end mill for socket head cap screw.
Op-10 Chamfer	De-burr all edges by chamfer milling with a center drill or chamfer mill.

Figure 29: Housing, Top Side Machining Operations

3 Job 2 Setup

Flip the part about the Y-axis and grip the finished walls of the part. Set the G54 XY from the center of the through hole created in Job 1. De-burr the hole before locating its center using dial indicator, coaxial indicator, or part probe. Set G54 Z the same way as Base, Job 2 (using 1-2-3 blocks) to the top of the finished face of the part bottom.

4 Job 2 Operations

Machine the part using the processes and sequence in Figure 30. Select appropriate tools, speeds, feeds and machining parameters.

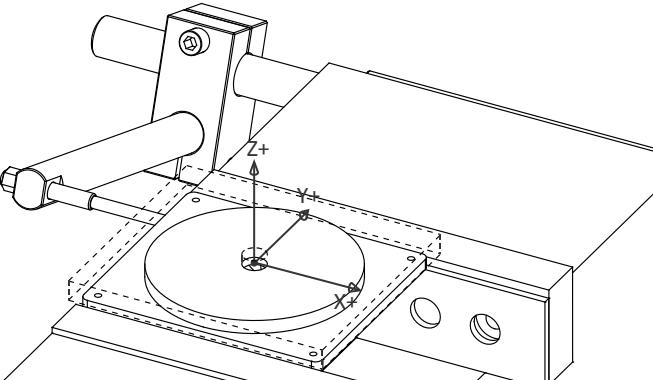
Housing: Job-2 Setup	
G55	
G55 Datum: XY00 is the center of the hole created in Job 1. Z0 is set at the face of the finished bottom side (Z.500 _(in)) from top of parallels.	
Op-1 Face	Finish part to thickness. Use light cuts to prevent part from being pulled out of vise.
Op-2 Contour OD	Rough and finish OD of round boss.
Op-3 Chamfer Mill	De-burr all edges.

Figure 30: Housing, Bottom Side Machining Parameters

Project 7: Fan Blades

This project teaches the following skills:

- Making and using soft jaws.
- Create feature to locate part for subsequent operation.
- 3D Roughing.
- 3D Finishing.

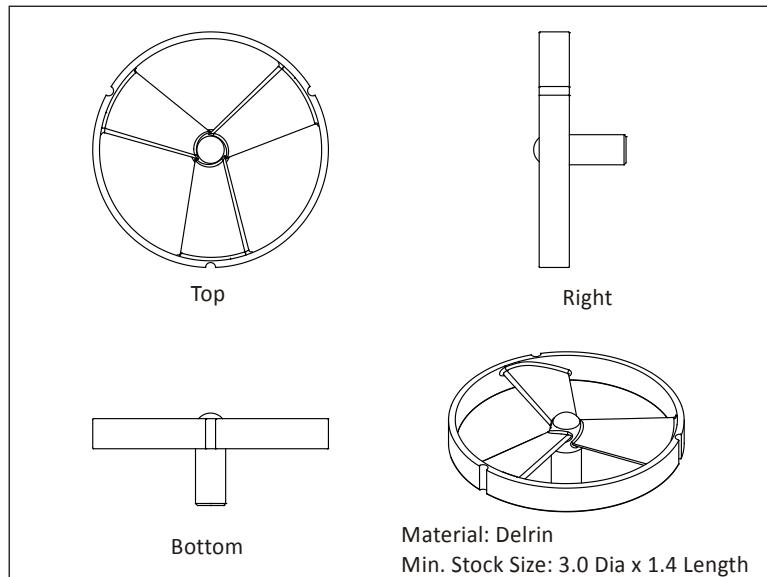
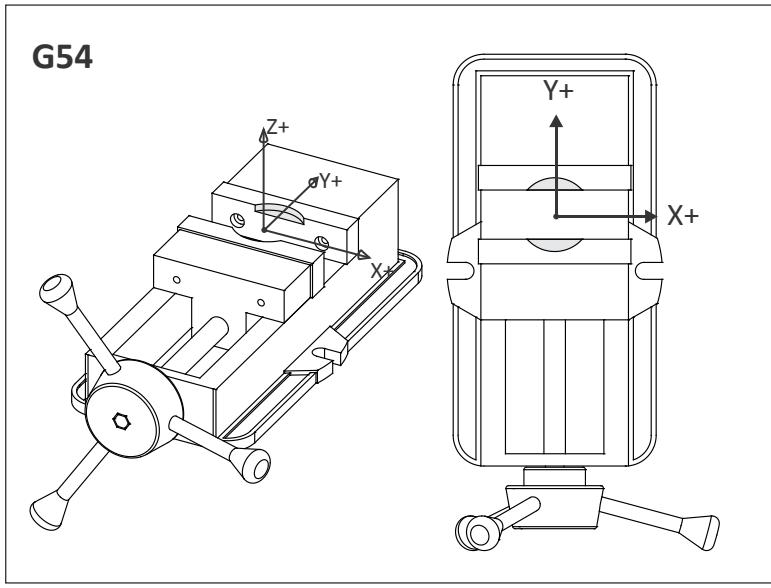


Figure 31: Project-7, Fan

1 Create Job-1 Soft Jaws Fixture

Cut a pocket into the aluminum jaws as shown in Figure 32. This is used to grip the sawed raw stock while machining the top side (G54).

Soft jaws for this part require advanced planning. Round stock is usually slightly larger than the size ordered, so either first turn it down on to the desired size on a lathe, or cut the jaws to match the actual material size. The jaws have to be cut deep enough to grip the part securely during machining. Cutting forces in Delrin are small, so a grip depth of .30 inches is adequate to secure this part.

**Figure 32: Job-1 Soft Jaws**

The XY datum is established by the pocket machined into the jaws. In other words; soft jaws both grip and set the datum for the part.

Before machining the jaws, place a bar (a vise hard jaw works well) between the vise jaws about 2.00 inches wide. This stabilizes the jaws during machining and sets the grip width.

One way to find the approximate location of the pocket center is to mark the center of the stock with a pencil, and set the stock on the top of the jaws. Load a small drill in the tool holder and jog the machine over the XY center of the stock and set this as the G54 X0Y0 position.

Machine the pocket using a spring pass to ensure the walls are straight. De-burr the top edge of the pocket by creating a .05 chamfer.

2 Set G54 Z Position

It is safest to shift the G54 Z upward so all rapid moves are +Z values. Because sawed stock varies in length typically by .02-.05in, set the G54 position to the top surface of the shortest stock material to be machined.

3 Machine Job 1 (Top Side)

Tighten the jaws firmly with a torque wrench, or mark the vise handle location after closing so the vise can always be tightened to the same pressure. Machine the blade using the processes in Figure 33.

Fan: Job-1 Setup

G54	Stock Material			
	Locating Hole (For Op2)			
G54 Datum: XY Same as soft jaw pocket center. Set G54 about .01 below the top of the shortest length of sawed stock.				
Op-1 Face .375 End Mill				
Tool (inch)	Speed (rpm)	Feed XY (ipm)	Feed Z (ipm)	
.375	4100	40.	20.	
Finish part to thickness. Use light cuts to prevent part from being pulled out of vise. Include enough rough passes to accommodate the tallest piece of stock to be machined.				
Op-2 Drill .125 Drill				
Tool (inch)	Speed (rpm)	Feed XY (ipm)	Feed Z (ipm)	
.125 Drill	4600	N/A	18.	
Drill the three holes around the perimeter of the fan ring, and the locating hole for Op2. The locating hole will be used to set the part orientation for Job 2 so that the top and bottom fan blades surfaces match precisely.				
Op-3 3D Rough .188 (3/16) End Mill				
Tool (in)	Speed (rpm)	Feed XY (ipm)	Feed Z (ipm)	
.188 (3/16)	8100.	80.	40.	
Rough the outside and inside surfaces of the fan ring to a depth of Z-.40. Leave stock of .005 on all surfaces.				
Op-4 3D Finish .125 (1/8) Ball Mill				
Tool (in)	Speed(rpm)	Feed XY (ipm)	Feed Z (ipm)	
.125 (1/8)	8.	8.	4.	
Pre-finish the inside surfaces of the fan to remove all scallop steps. Use a spiral or circular path and a stepover of .09. Leave .005 stock on all surfaces.				
Op-5 3D Finish .093 (3/32) Ball Mill				
Tool (in)	Speed (rpm)	Feed XY (ipm)	Feed Z (ipm)	
.093 (3/32)	10,000	75.	30.	
Finish the inside surfaces of the fan. Use a spiral or circular path and a stepover of .01.				
Op-6 3D Pencil .093 (3/32) Ball Mill				
Tool (in)	Speed (rpm)	Feed XY (ipm)	Feed Z (ipm)	
.093 (3/32)	10,000	75.	30.	
Pencil mill the inside surfaces to remove any scallops remaining on the inside corners. Use a stepover of .01(in).				
Op-7 2D Contour .188 (3/16) End Mill				
Tool (in)	Speed (rpm)	Feed XY (ipm)	Feed Z (ipm)	
.188 (3/16)	8100.	80.	40.	
Finish the OD of the part to a depth of Z-.4.				
Op-8 2D Chamfer .25 Center Drill				
Tool (in)	Speed (rpm)	Feed XY (ipm)	Feed Z (ipm)	
.25 Center Drill	6000.	24.	12.	
Create the chamfers on the top of the ring. Use line/arc lead in/out and take two passes.				

Figure 33: Fan, Job 1 Toolpaths

4 Create Job-2 Soft Jaws Fixture

Replace the soft jaws with new blanks or re-cut the previous jaws. Set up and cut the jaw pockets for Job-2 exactly like for Job-1. De-burr the top edge of the pocket by creating a $.01\text{in}$ chamfer.

These jaws include a $.125\text{in}$ diameter hole used to set the rotary index position of the part for the second operation. This hole should extend about $.25\text{in}$ into the soft jaw step.

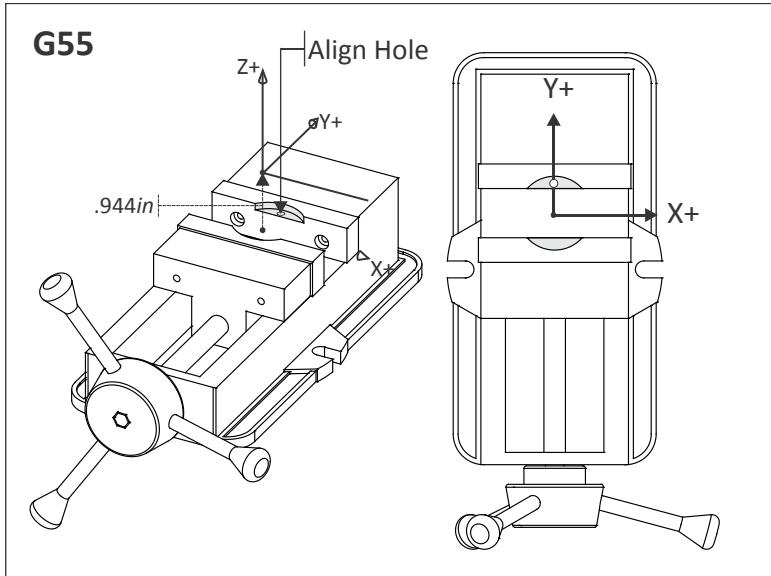


Figure 34: Soft Jaws for Job-2

5 Set G55 Z Position

Set the G55 Z Position off the floor of the soft jaw pocket, then shift it upward $.944\text{in}$ above this surface (Distance from face of fan ring to bottom of the part).

G55 Z could be set to the face of the soft jaw pocket as long as the top face of the ring is set as the G55 datum in the CAM system. Just be sure to set the clearance, rapid, and feed heights so the tools clear the part and stock during machining.

6 Load Part into Job-2 Fixture

Lightly tighten the vise jaws so part still rotates in fixture.

Next, insert a $.125\text{in}$ diameter gage pin (or a $1/8\text{in}$ drill upside-down) through the locating hole in the stock (drilled in Job 1), and into the hole in the fixture. Use this to align the part so the surfaces machined in Job 1 line up perfectly with the surfaces machine in Job 2. Securely clamp the jaws. **Remove the pin or drill before machining.**

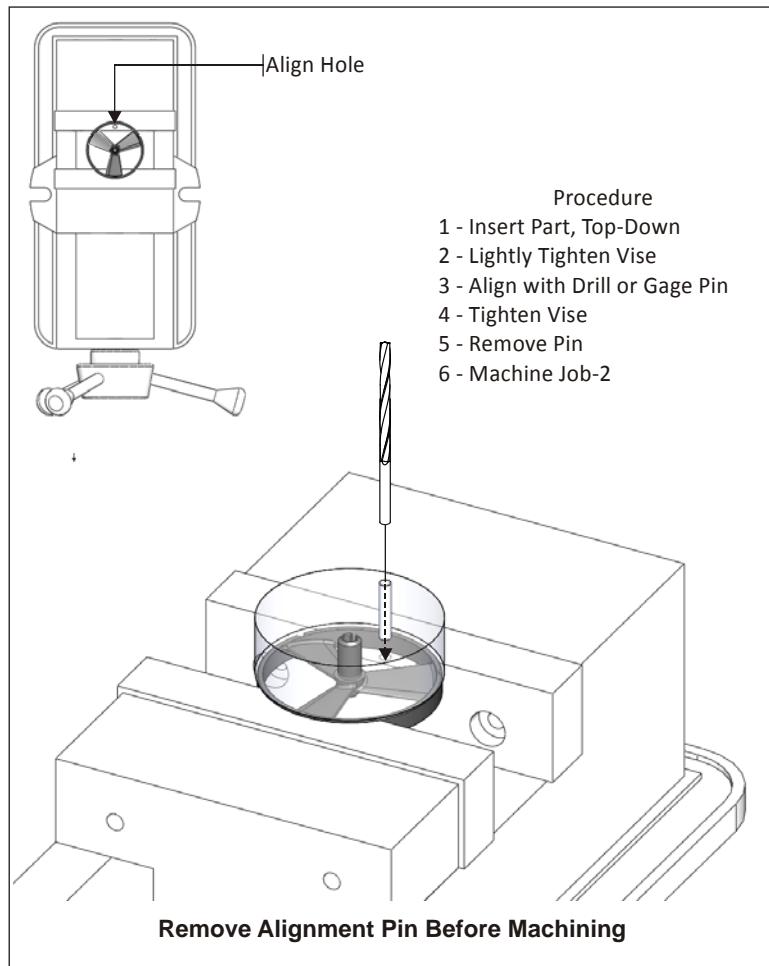


Figure 35: Use Drill or gage pin to Align Part

7 Machine Job-2 (Bottom)

Machine the bottom of the fan using the operations listed in Figure 37. Select the appropriate tools and machining parameters.

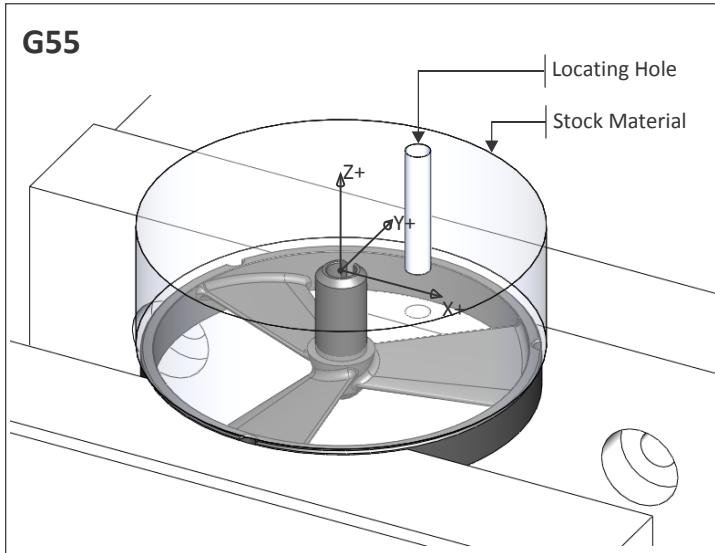


Figure 36: Job 2 Setup, Fan

Fan: Job-2 Setup	
G55 Datum: XY Same as soft jaw pocket center. Set G55 from bottom of pocket to top of finished part (Z+.994 inches).	
Op-1 Face	Finish part to thickness. Use light cuts to prevent part from being pulled out of vise.
Op-2 Drill	Peck drill hole in center of spindle.
Op-3 3D Rough	Rough the bottom of the part leaving stock for finish passes.
Op-4 2D Contour	Finish the OD of spindle.
Op-5 3D Finish	Pre-finish the bottom blade surfaces leaving stock for finish machining.
Op-6 3D Finish	Finish the bottom blade surfaces.
Op-7 2D Contour	Use a ball mill to machine the slot in the spindle.
Op-8 2D Contour	Finish the bottom of the blade shroud.
Op-8 3D Pencil	Pencil mill to remove scallops in the fillets between the blades and housing and spindle.
Op-10 Chamfer Mill	Create the chamfers on the edges of the housing.

Figure 37: Fan, Job 2 Toolpaths

Project 8: Clamp

Clamps are a very common work holding device. Many sizes and styles of clamps are commercially available. This project shows how to machine custom size aluminum clamps that will be used to hold the part in Project 9. This project teaches the following skills:

- How to create a custom clamp.
- How to create multiple parts from a single piece of stock.
- How to use 3D machining to create 2D features instead of using corner rounding tools.

Figure 38 shows a typical custom clamp. The **pivot** rests on the fixture plate and allows the clamp to freely rotate. The **slot** is slightly wider than the bolt used to secure the clamp and allows the clamp to be positioned. The **pad** that rests on the stock is offset by the thickness of the material to be gripped.

If the part being held by the clamps is aluminum, use aluminum clamps to reduce marring of the part. If using steel clamps to hold softer metals, consider inserting an aluminum pad between the pad and the part.

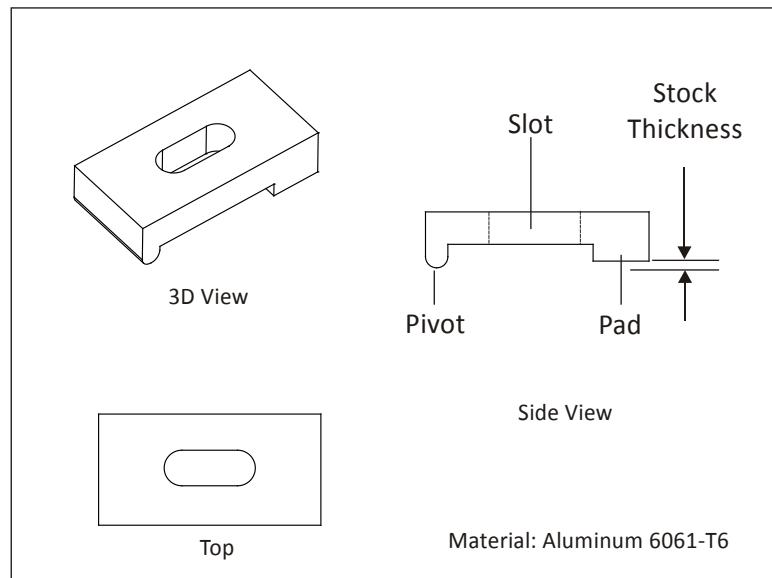


Figure 38: Custom Clamp Features

1 Job 1 Setup

Figure 39 below shows how all clamps can be machined from a single bar of stock. Machine the pivot radius using a corner round tool, or 3D surface machine it with a flat cutter.

2 Job 1 Operations

Once the part has been completed as shown in this illustration, flip it over in the vise and machine the top surface to free the parts.

Fan: Job-2 Setup	
G54	
Op-1 Face	Face top of pad, slot and radii.
Op-2 2D Contour	Cut around perimeter of all parts.
Op-3 Slot Mill	Machine slots.
Op-4 3D Mill	3D finish the radii. Surface finish is not important.
Op-5 Face	Flip part over and face to free all clamps from bar stock.

Figure 39: Clamps, Bottom Side Machining Parameters

Project 9: Fixture Plate

This project teaches the following skills:

- How to create a fixture plate.
- How to use clamps.
- Counterboring.

A fixture plate is a work holding device used to hold thin parts that cannot be clamped in a vise. In prototyping, the plate is usually made of aluminum and is resurfaced and reused. Small fixture plates can be held in a vise and larger plates are bolted or clamped to the table.

Figure 40 below shows an aluminum plate fastened directly to the machine table with a toe clamp: a low profile clamp that grips the edge of the plate and pushes it against the table. Once the plate is fastened to the table, face mill the plate to ensure it is flat.

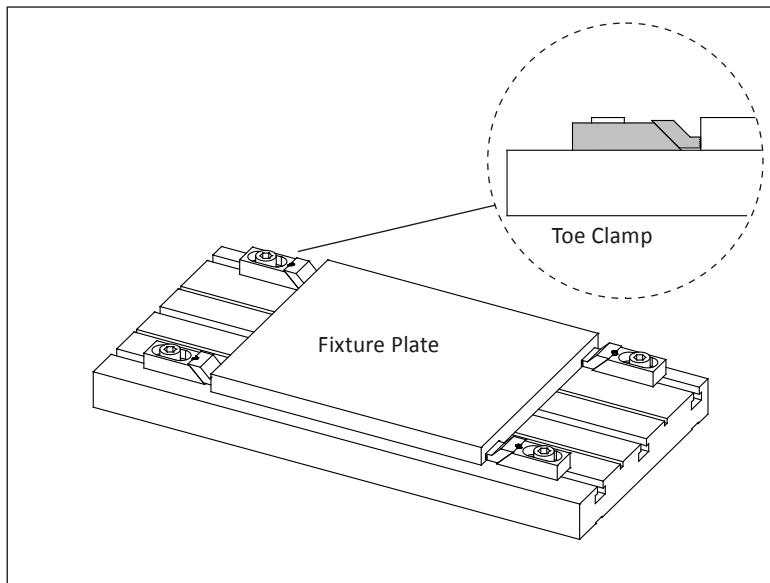


Figure 40: Fixture Plate

Use caution when drilling holes in the fixture plate to not drill into the machine table.

Tips for Fixture Plates:

- After the plate is secured, machine the face flat to ensure it is parallel with the machine table. If the plate is reused, the face will likely have to be machined again.
- It is almost impossible to re-align a fixture plate after it has been removed from the vise or machine table. If you need to reuse the plate to make the same parts, machine all sides flat and square first, and note the datum location on the plate by engraving or stamping letters indicating the datum values on the plate.
- Eventually threads in aluminum plates will wear, so use a steel or Helicoil® insert for higher production plates.
- Never rely on threads alone to locate a part precisely. Use dowel pins or shoulder bolts to ensure accurate location when machining operations on the opposite side.
- Be creative. Use the components above in whatever combination works best for the job. This project shows moving clamps between operations and installing bolts in areas that will be later cut away. Both are techniques that are often used in work-holding.

Figure 41 shows a fixture plate used to make Project 10: French Curve. This example includes many of the components used in fixture plates.

Clamps hold the plate down. Clamps are versatile, and can be moved between operations to grip the part in different places for different machining paths.

Dowel pins are used to precisely locate the stock or part. To ease pin removal, grind a small flat on the side of the pin. This allows air to fill the hole as the pin is extracted. Without a vent, the pin will be difficult or impossible to remove once installed.

Bolts are used to hold down the part after it is clamped and drilled.

Fixture Plate: Job-1 Setup	
G54	
Datum:	Upper-left corner of fixture plate.
Op-1 CTR Drill	Center drill all holes.
Op-2 Drill	Drill .25 holes (3X) for dowel pins. Drill these just deep enough so the dowel pins extend about .25 inches above the plate face.
Op-3 Drill	Drill the holes for the #6-40 bolts that will be used to hold the part down while the OD is machined.
Op-4 Drill	Drill the holes for the ¼-20 bolts used to secure the custom clamps.
Op-5 Tap	Tap the #6-40 holes.
Op-6 Tap	Tap the ¼-20 holes.

Figure 41: Fixture Plate Machining Parameters

Project 10: French Curve

This project teaches the following skills:

- Using fixture plate.
- Using and moving bolts and clamps.
- Spline machining.

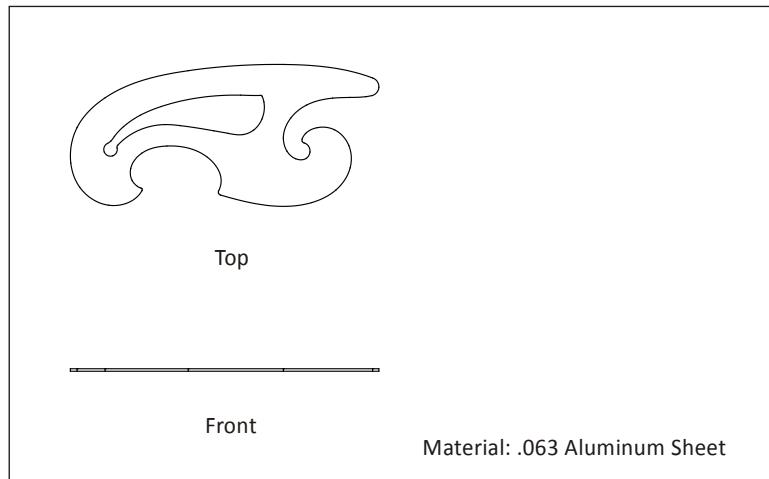


Figure 42: French Curve

Plate parts can present unique work holding challenges. Thin and flexible material tends to be lifted and torn by the cutter. This exercise uses the fixture plate created in Project 9 and works with parts that have good rigidity.

This same process can be used with very thin and flexible parts that would normally require stamping or laser cutting by sandwiching the material between a backing plate and the fixture plate, as shown in Figure 43 below.

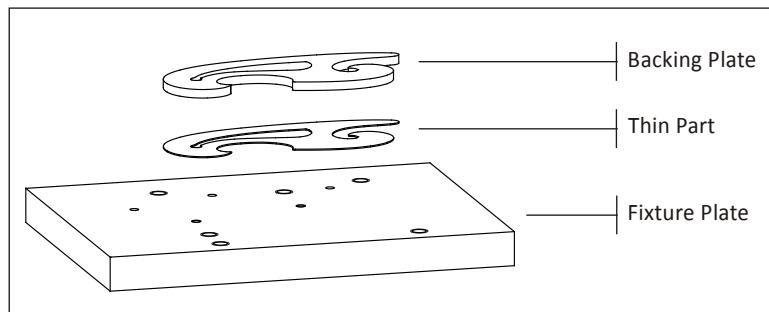


Figure 43: Thin Part with Backing Plate (Finished)

The following instructions show how to use the fixture plate created in Project 9 to machine a sheet metal part.

1 Clamp Part Into Fixture Plate

Use the dowel pins to locate the stock material as shown in Figure 44 and tighten the clamp bolts firmly.

If using steel clamps on aluminum or other soft material, adjust clamp pressure carefully and consider placing a piece of aluminum between the clamp and part to prevent marring.

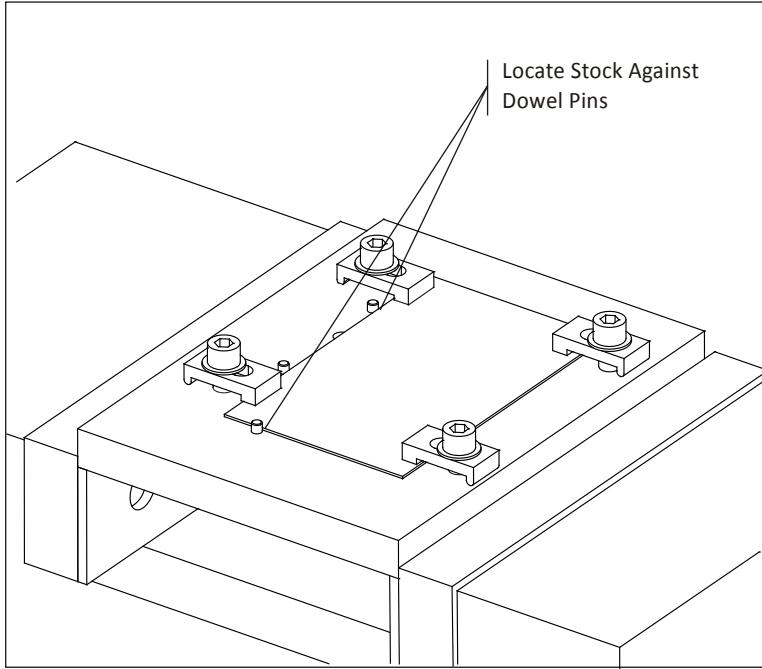


Figure 44: Clamp Part

2 Remove Dowel Pins

Any contact between a hardened dowel pin and tool will destroy the tool. Remove the dowel after clamping if there is any chance the tool could come in contact with it.

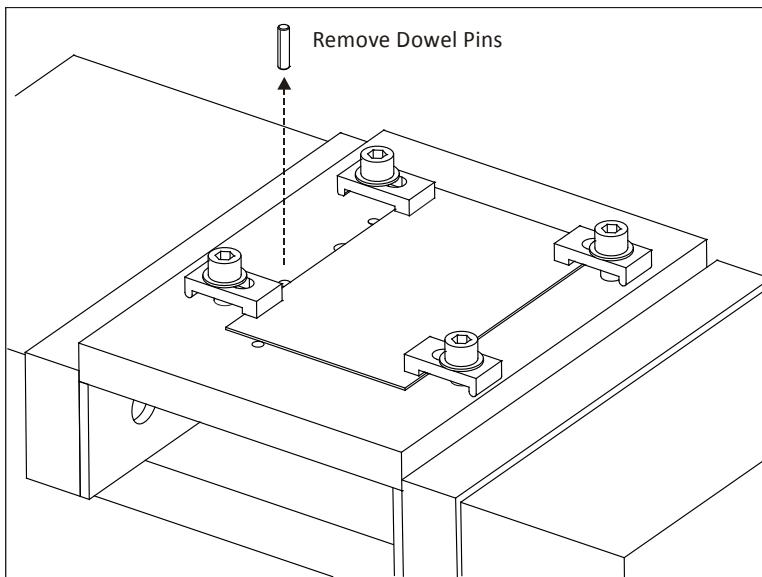


Figure 45: Remove Dowel Pins

3 Drill Hold-Down Holes

Drill Holes for the #6-40 screws that will secure the part after the stock being gripped by the clamps is cut away. The drill depth should be just enough so the shoulder clears the bottom of the part, but does not cut deep into the threads in the fixture plate.

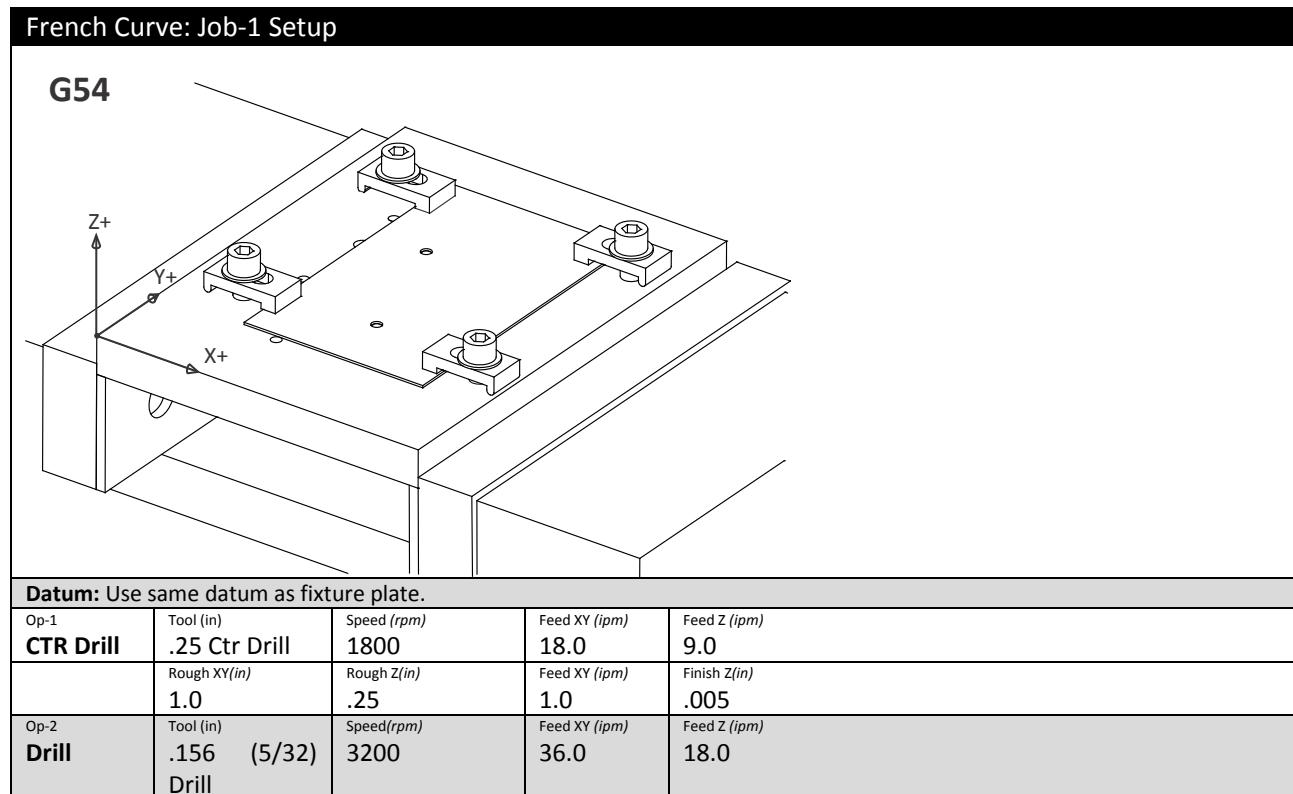


Figure 46: Fixture Plate Job 1

4 Install Hold Down Bolts

Install bolts. Remove the clamps if the tool will contact them, otherwise leave them in place so the scrap material is gripped securely after as it is cut away.

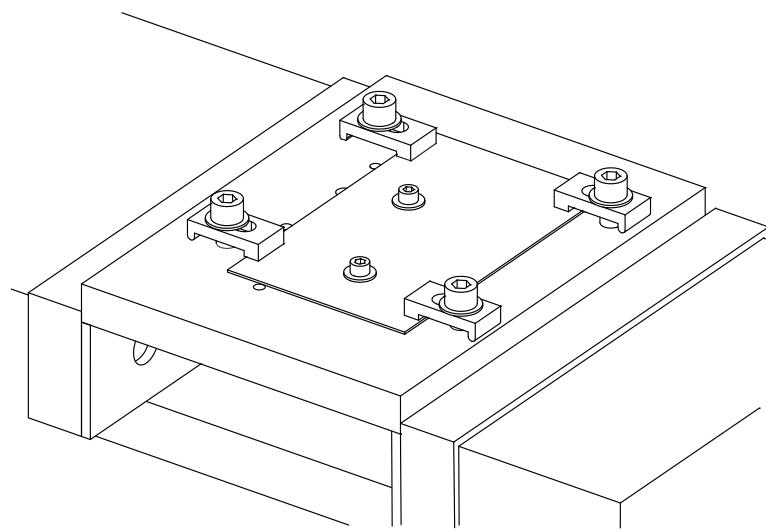


Figure 47: Install Bolts

5 Machine OD

Cut the outside profile of the part as shown in Figure 48.

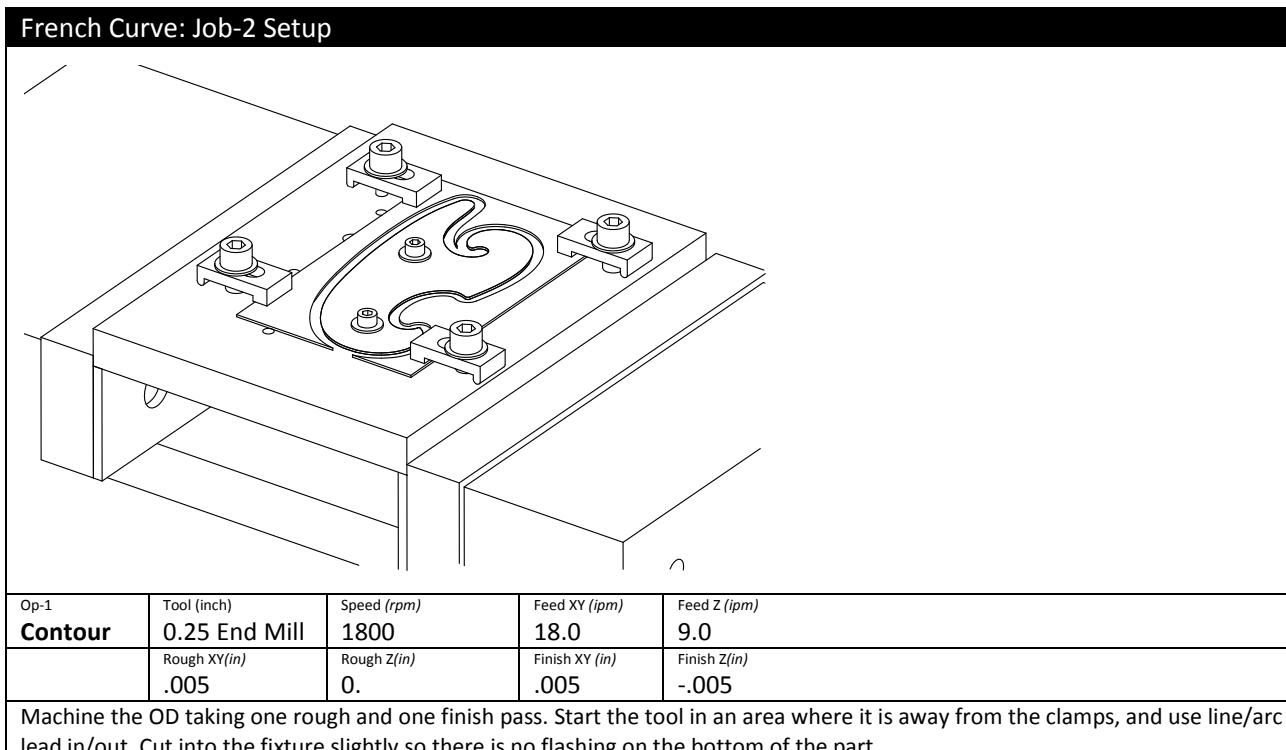


Figure 48: Fixture Plate, OD Machining Parameters

6 Move Clamps, Remove bolts

First move two of the clamps to the alternate positions shown so they secure the OD of the part. Then remove the hold down bolts so the ID of the part can be machined.

Be careful when tightening the clamp bolts. The part is thin and excess pressure might cause the part to deform. Of course, insufficient gripping force might allow the part to shift under the cutting forces and ruin the part. Using the custom clamps (Project 8) machined for this part maximize the grip surface area.

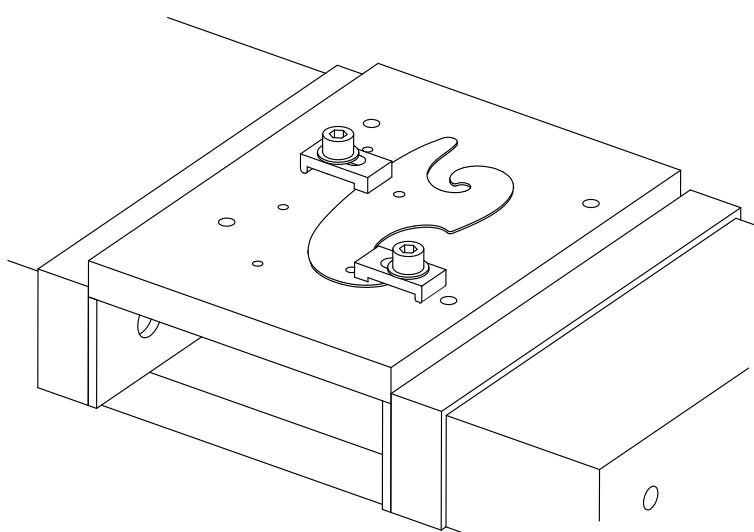


Figure 49: Move Clamps

7 Machine ID

Machine the ID Profile to complete the part.

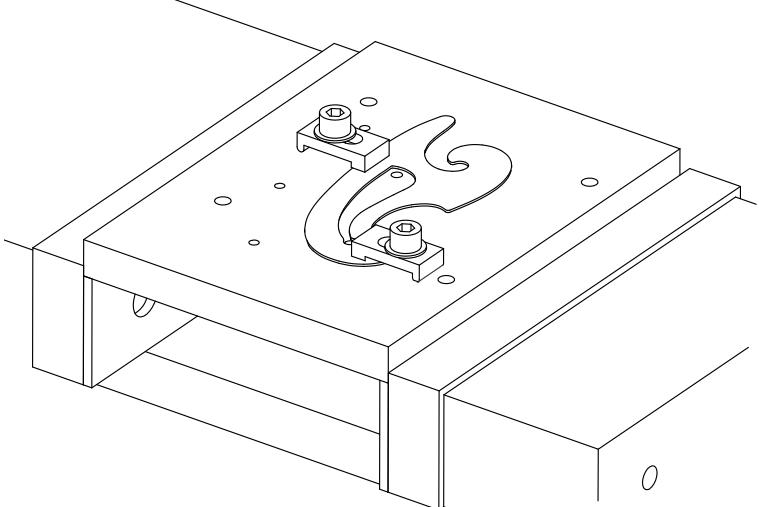
French Curve: Job-3 Setup				
				
Op-1 Contour	Tool (in) 0.25 End Mill	Speed (rpm) 1800	Feed XY (ipm) 18.0	Feed Z (ipm) 9.0
	Rough XY(in) 1.0	Rough Z(in) .25	Feed XY (ipm) 1.0	Finish Z(in) .005
	Stock to Leave XY(in) 0.0	Stock to Leave Z(in) 0.0		

Figure 50: Fixture Plate, ID Machining Parameters

To observe the effects of data starvation, set the tool length offset so the tool runs above the part. Run the program, increasing feed rate until the machine begins to shudder. Notice that the actual machine feed rate is less than the programmed feed rate.

The feed rate at which this occurs depends on the block execution time of the machine tool. Reduce or eliminate data starving by reducing the feed rate at the machine, or by using the tool path filter function in the CAD/CAM software.

Project 11: Honeycomb

This project teaches the following skills:

- Thin wall machining.
- Small corner radii machining.

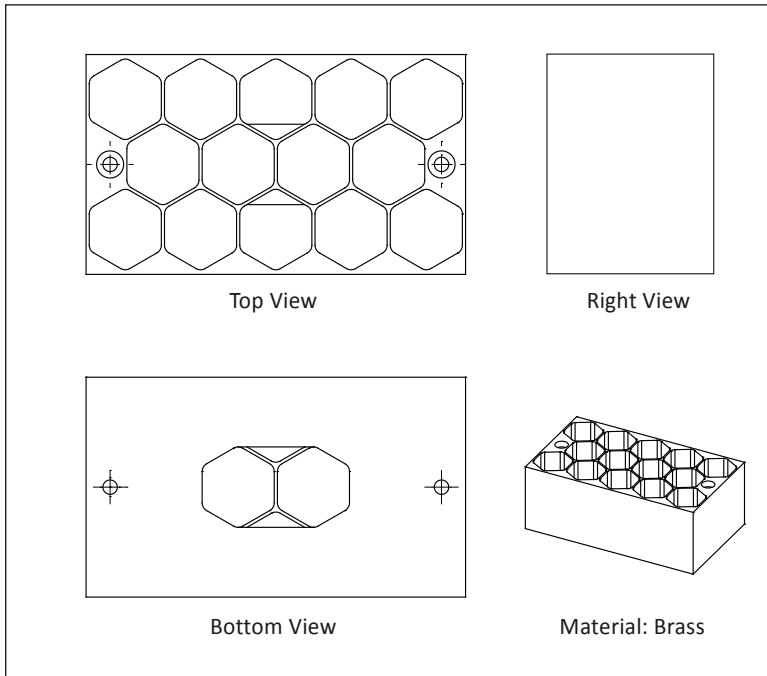


Figure 51: Honeycomb

1 Clamp Part In Vise

Clamp the sawed stock in vise as shown in Figure 52. Set the G54 datum off the upper left corner of the stock. Be sure to leave enough stock so the face and outside perimeter tool paths remove material.

Honeycomb: Job-1 Setup				
G54				
Datum:	Upper-left corner of stock.			
Op-1 Face	Tool (inch) 0.25 End Mill	Speed (rpm) 1800	Feed XY (ipm) 18.0	Feed Z (ipm) 9.0
	Rough XY 1.0	Rough Z .25	Finish XY 1.0	Finish Z .005
	Stock to Leave XY 0.0	Stock to Leave Z 0.0		

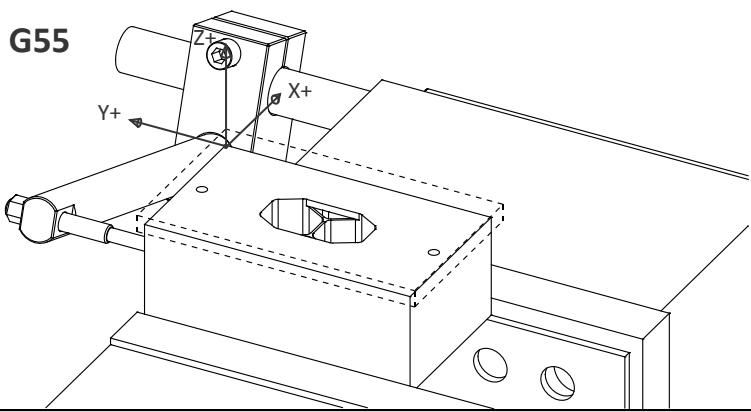
Op-2 Ctr Drill	Tool (in) 0.25 Ctr Drill	Speed (rpm) 1800	Feed XY (ipm) 18.0	Feed Z (ipm) 9.0
Op-3 Drill	Tool (in) 0.25 Drill	Speed (rpm) 1800	Feed XY (ipm) 18.0	Feed Z (ipm) 9.0
Op-4 Pocket	Tool (inch) 0.25 End Mill	Speed (rpm) 1800	Feed XY (ipm) 18.0	Feed Z (ipm) 9.0
	Rough XY(in) 1.0	Rough Z(in) .25	Finish XY(in) 1.0	Finish Z(in) .005
	Stock to Leave XY(in) 0.0	Stock to Leave Z(in) 0.0		
Op-5 Pocket	Tool (in) 0.25 End Mill	Speed (rpm) 1800	Feed XY (ipm) 18.0	Feed Z (ipm) 9.0
	Rough XY(in) 1.0	Rough Z(in) .25	Finish XY(in) 1.0	Finish Z(in) .005
	Stock to Leave XY(in) 0.0	Stock to Leave Z(in) 0.0		
Op-6 Remachine	Tool (in) 0.25 End Mill	Speed (rpm) 1800	Feed XY (ipm) 18.0	Feed Z (ipm) 9.0
	Rough XY(in) 1.0	Rough Z(in) .25	Finish XY(in) 1.0	Finish Z(in) .005
	Stock to Leave XY(in) 0.0	Stock to Leave Z(in) 0.0		

Figure 52: Honeycomb, Top Side Machining Parameters

2 Flip Part In Vise

Flip part about Y-axis, and set the datum as the upper-left corner of the finished bottom face. Use the method described in Project 5 (Steps 3-4) to set G55 for this part.

Honeycomb: Job-2 Setup



Datum: Upper-left corner of finished top face.

Op-1 Face	Tool (inch) 0.25 End Mill	Speed (rpm) 1800	Feed XY (ipm) 18.0	Feed Z (ipm) 9.0
	Rough XY(in) 1.0	Rough Z(in) .25	Finish XY(in) 1.0	Finish Z(in) .005
	Stock to Leave XY(in) 0.0	Stock to Leave Z(in) 0.0		
Op-1 Contour	Tool (inch) 0.25 End Mill	Speed (rpm) 1800	Feed XY (ipm) 18.0	Feed Z (ipm) 9.0
	Rough XY(in) 1.0	Rough Z(in) .25	Finish XY(in) 1.0	Finish Z(in) .005
	Stock to Leave XY(in) 0.0	Stock to Leave Z(in) 0.0		

Figure 53: Honeycomb Job 2

Project 12: Bracket

This project teaches the following skills:

- How to use various types of clamps.
- How to rearrange clamps to avoid machining operations.
- How to use an angle plate.
- Planning ahead taking into consideration part geometry and changes in rigidity as material is removed.

Almost every part presents unique work holding challenges. Part geometry, material type, wall thickness, and features all influence work holding.

Parts like this present multiple challenges. Like a chess player, a machinist has to think ahead several moves to visualize what the part will look like each step of the way. Before starting this project, it is important to consider machining methods, sequencing, work holding, and potential problems.

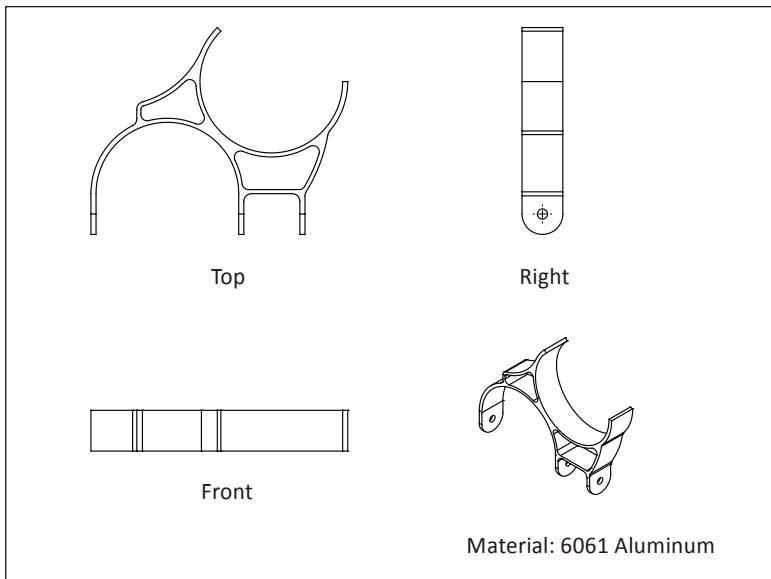


Figure 54: Project-11, Bracket

1 Clamp Part to Table

Use clamps to secure the part to the table as shown in Figure 55 below. Flanges will be cut on the ends of the stock to hold and locate the part in subsequent operations. Plan ahead. Use stock thick enough to form rigid flanges and face the part on both sides. Locate clamps clear of tool paths and set rapid heights to easily clear the clamps.

Many styles of clamps are available. The type shown here uses Step Blocks to set the clamp at the same level as the top of stock. Machining on this side of the part produces significant cutting pressures. Because the clamps rest on material that will be later cut away, there is no problem with marring the finished part, so the bolts can be tightened very securely to prevent the part from slipping.

Once the part is clamped to the table, it must not be moved until all machining on this side are complete.

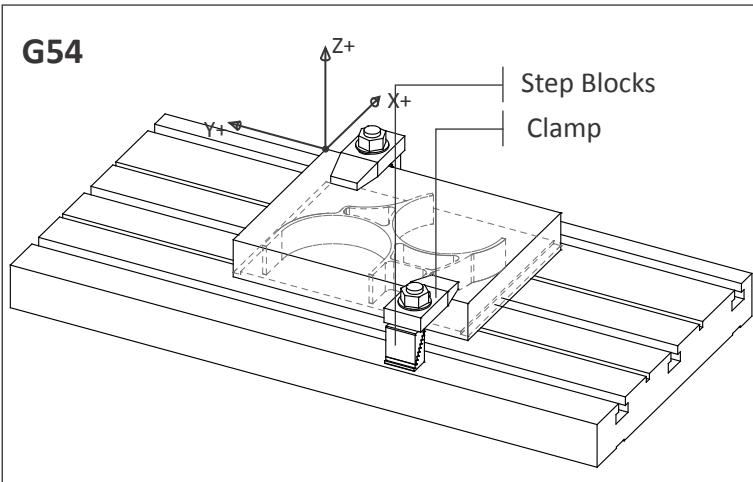


Figure 55: Bracket, Job-1 Setup

2 Machine Flanges

Machine flanges on the part ends as shown in Figure 56. These flanges serve two purposes. The first is to grip the part while the top side is machined. The second is to level the part when machining the left and right sides.

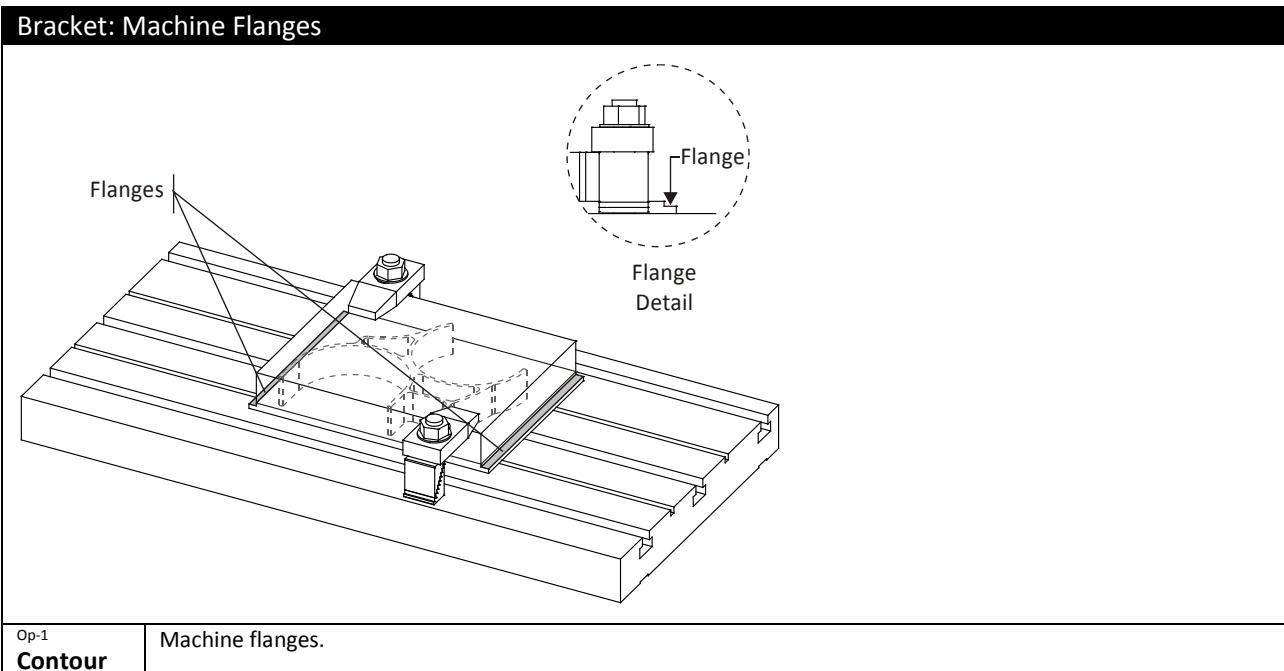


Figure 56: Bracket, Machine Flanges

3 Install Edge Clamps

Being careful not to shift the part at all, add clamps to the flanges. Use low profile clamps and plan operations carefully to ensure the tool does not contact the clamps or create excessive cutting pressure that might shift the part.

Use as many clamps as necessary to secure the part. Many styles and size of clamps are commercially available, or you can fabricate your own. Most cutting pressure is in the XY plane, though the plate must be held against the table to prevent the part from bending as material is removed. Test to see if the part has lifted after machining by trying to slide a piece of notebook paper or feeler gauge between the table and part bottom. If the flange is thick enough and the clamping is good, the part will lie flat against the table after machining the top.

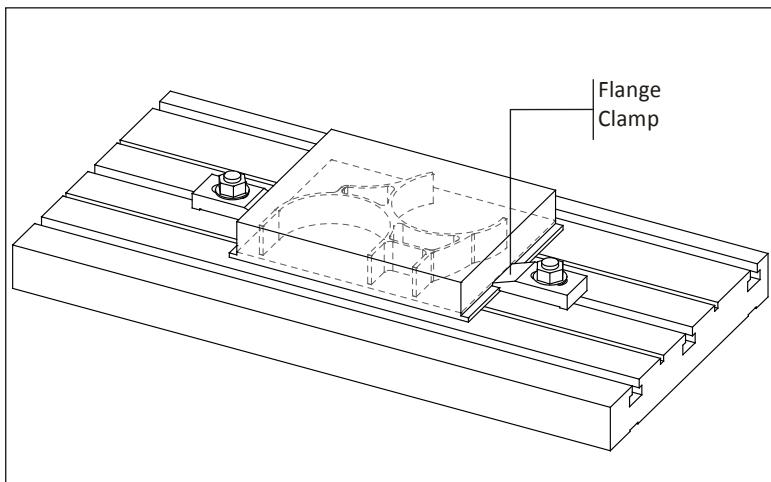


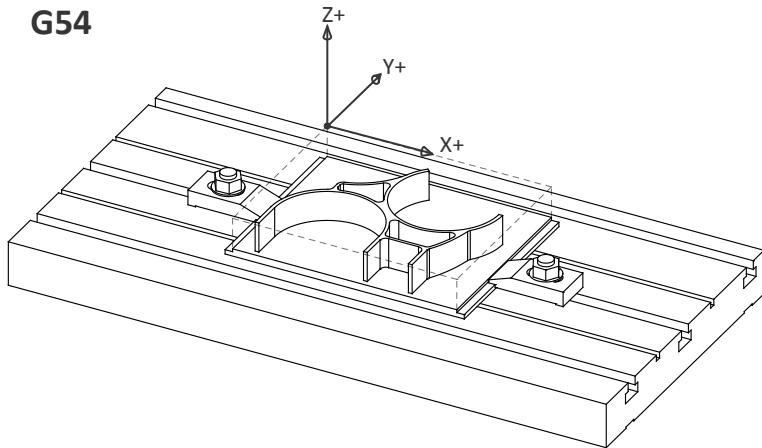
Figure 57: Install Flange Clamps

4 Machine Top

Face, pocket, and finish this side complete. Cut the perimeter of the part .010 inches below the profile to ensure no flashing remains after the bottom side of the part is faced off in the last operations.

Bracket: Job 1

G54



Datum: Same as part datum.

Op-1 Face	Face part.
Op-2 2D Contour	Rough and finish OD of part. Machine at least .005 below the bottom profile. Avoid clamps.
Op-3 Pocket	Rough and finish pockets.

Figure 58: Bracket, Machine Top Side

5 Set Up Angle Plate

An angle plate is a sturdy precision ground plate that is used to set parts on edge.

The next operation machines the holes and fillets on the right side. (This is done before finishing the bottom so the material helps support the part from bending while machining the ends.)

Bolt and align the angle plate to the table. Secure the part with toggle clamps and use a Machinists Square to level the part. Check by running a dial indicator across the flange as shown. This angle is critical, so make the part as level as possible.

The XY Datum can be found by picking up the reference point shown in Figure 59. Notice that the Y-datum is established by locating the finished part top against the angle plate.

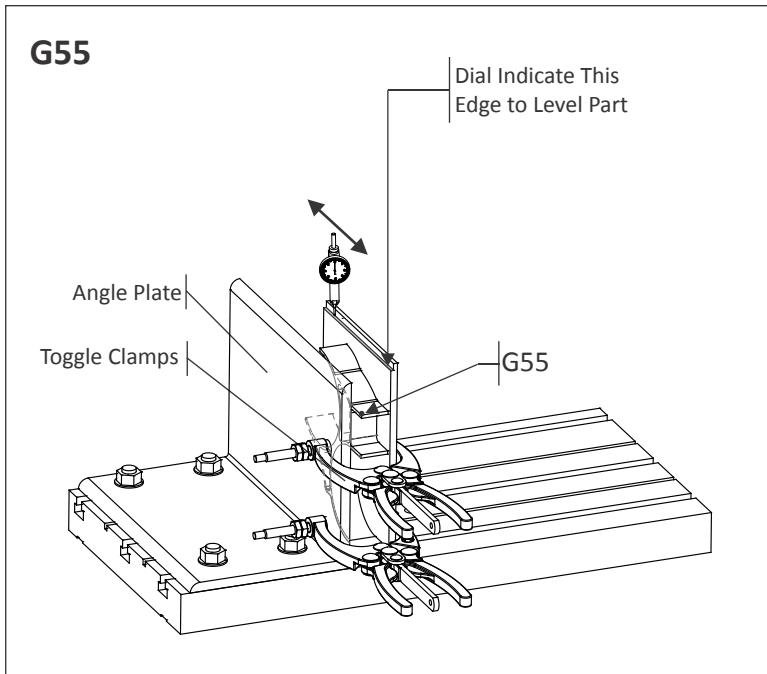


Figure 59: Angle Plate Setup

6 Machine Clearance For Job 2 Cuts

Carefully machine (or cut on a band saw) the part stock as shown in Figure 60 to provide tool clearance for the next operations. Use light cuts because heavy cuts could cause the part to shift. The edge being cut away is thin and not firmly supported —a situation prone to resonance. If the tool starts to chatter, slow feed rate and spindle speed.

The setup shown here is a tradeoff between tool clearance and work rigidity. The less the part hangs off the angle plate the more rigid the setup, but this reduces tool clearance needed to machine the radii.

As always, do not run the program unless certain the setup is stable and safe. Use the simulation and collision checking features of the CAD/CAM software and check for tool and holder interference with the remaining stock and angle plate.

This part is a good illustration of the difference between prototype and production manufacturing. Large production quantities might justify the cost of more complex fixtures that would allow the part to be easily clamped.

A rotary 4th axis might be used so that the part could be automatically reposition –eliminating several setups and improving the accuracy of the holes.

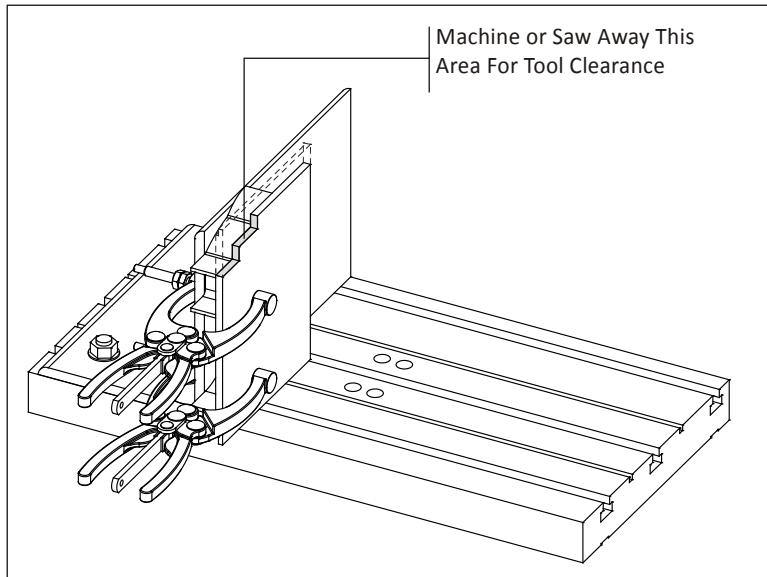


Figure 60: Bracket, Machine Tool Clearance

7 Drill Holes, Cut Radii

Center drill and drill the holes in the ears, and then contour the radii. Take small cuts and use the smallest recommended chip load to prevent tool chatter or deforming the part.

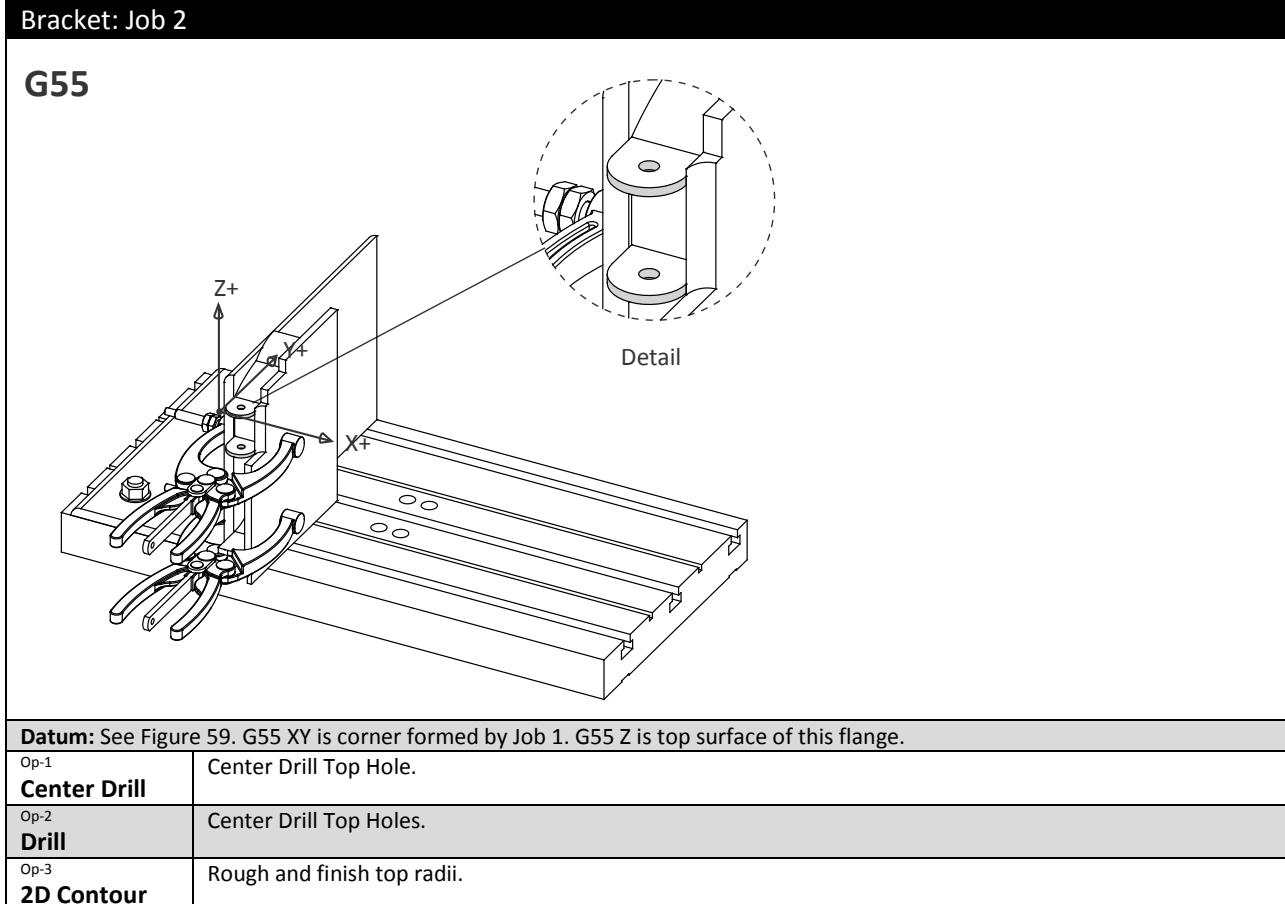


Figure 61: Bracket, Right Side

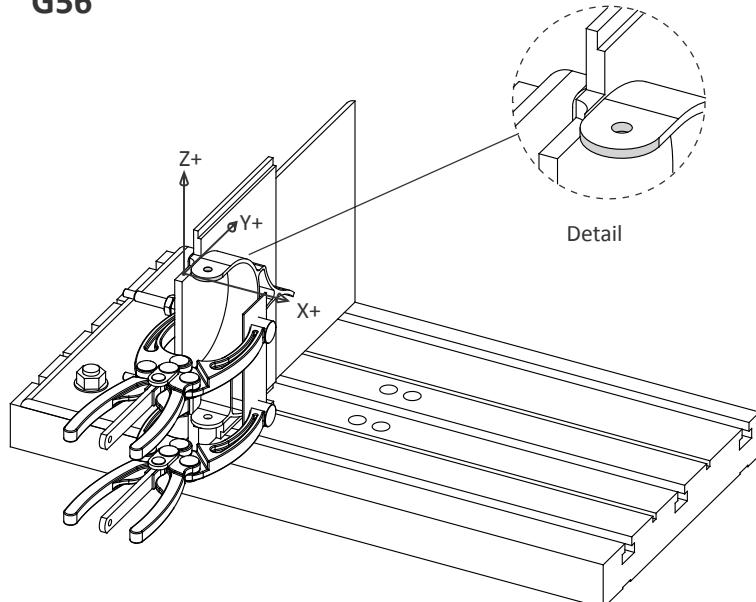
8 Drill Holes, Cut Radii, Left Side

Rotate the part 180 degrees about the Y-axis as shown in Figure 62 below. Use the same procedures as in Steps 5 thru 7 to level the part, find the datum, remove material for tool and holder clearance, and then machine the hole and radius.

This is a tricky setup because the piece is flipped, has deformed during machining, and the hole must be located precisely to align with the holes created in the previous operation. Do your best work and check your datum carefully. Scrapping the part at this step would be costly.

Bracket: Job-3

G56



Datum: Pick up G56 XY from corner of flange as in Step 7, Figure 61.

Op-1 Center Drill	Center Drill Top Hole.
Op-2 Drill	Center Drill Top Holes.
Op-3 2D Contour	Rough and finish top radii.

Figure 62: Bracket, Left Side

9 Face Bottom Ends

Set the part on the table face side down. Clamp in the center as shown in Figure 63.

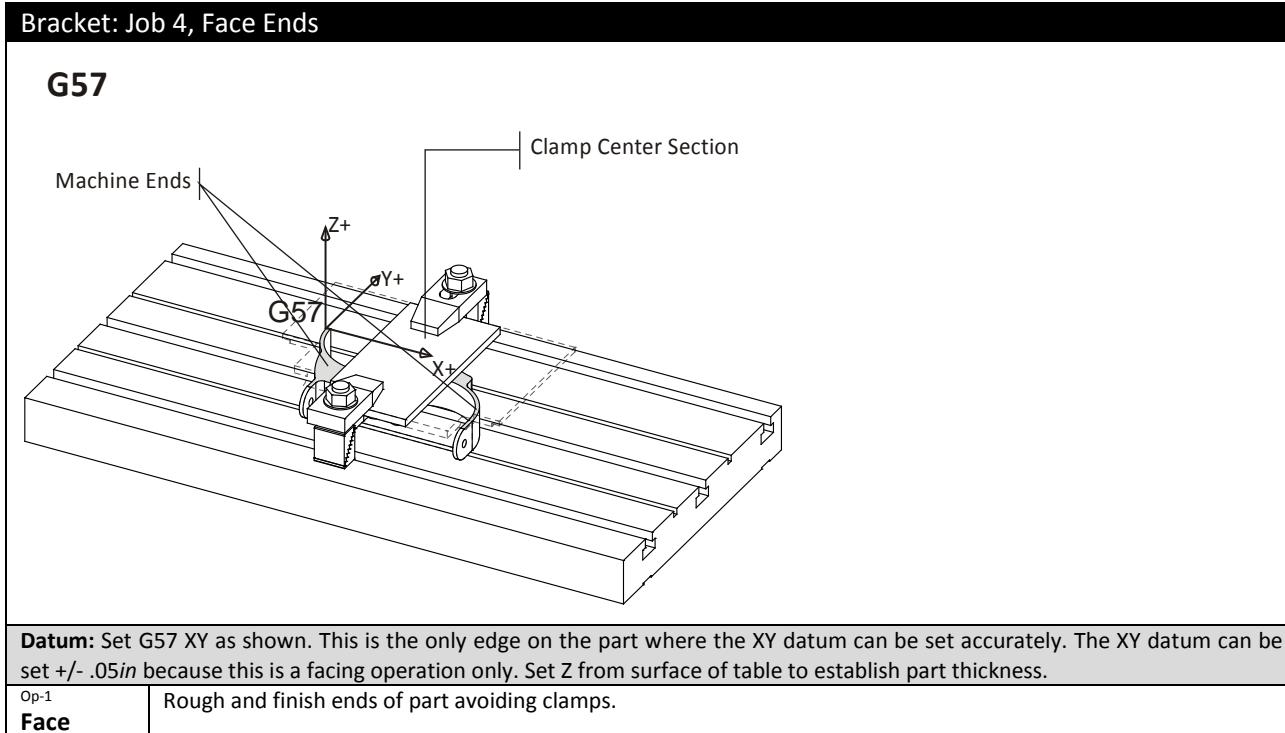


Figure 63: Bracket, Face Ends

10 Face Center Area

Install on the ends of the part as shown in Figure 64. Place aluminum pads under the clamps so they won't scratch or damage the part. Machine the center section to complete the part.

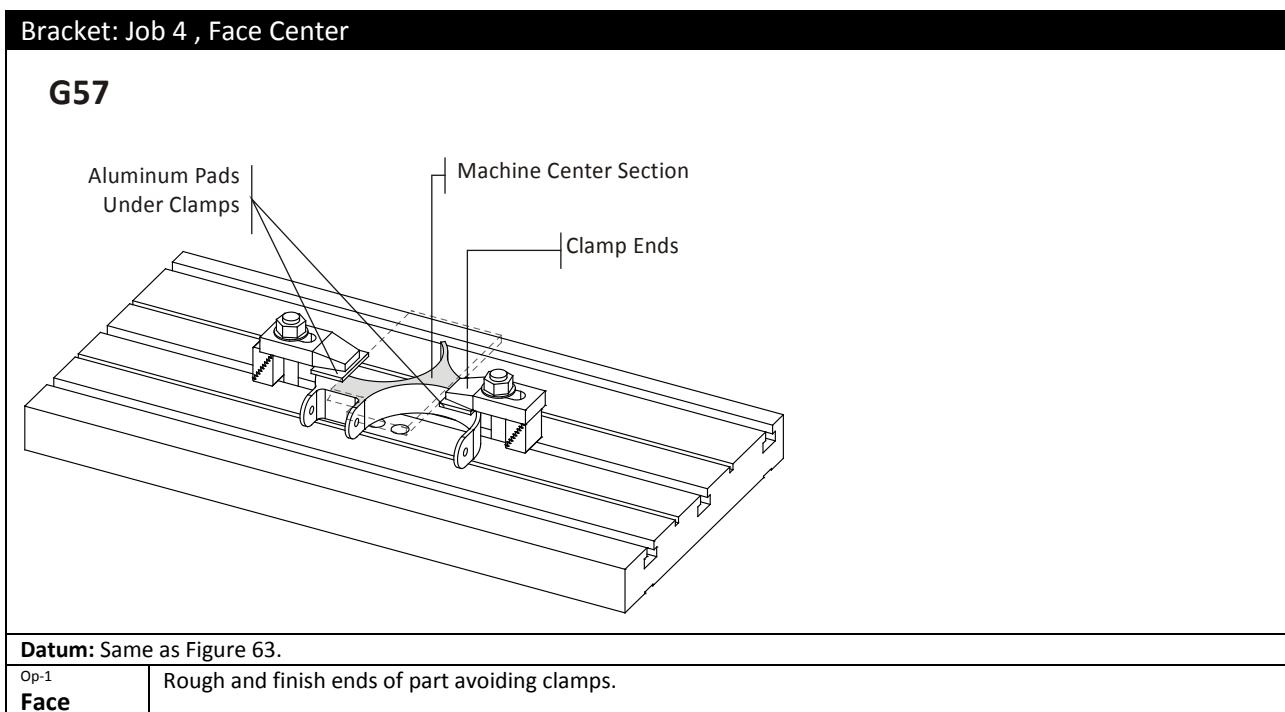


Figure 64: Bracket, Face Center

Project 13: Microwave Housing

This project teaches the following skills:

- Machining ramps.
- 2D REST Milling.
- Clamping direct to the table or sub plate.
- Using flange to support part during machining.
- Using dowel pins or shoulder bolts to precisely locate part.

This project is designed with features on the top and bottom that must match perfectly. Careful planning and setup is required to machine the part correctly, and avoid drilling into the table.

Large plate parts like this are usually clamped to the table because they will not fit in a standard vise, and vise pressure may cause the part to deform as material is removed. Clamping downward, rather than squeezing in a vise, prevents warping.

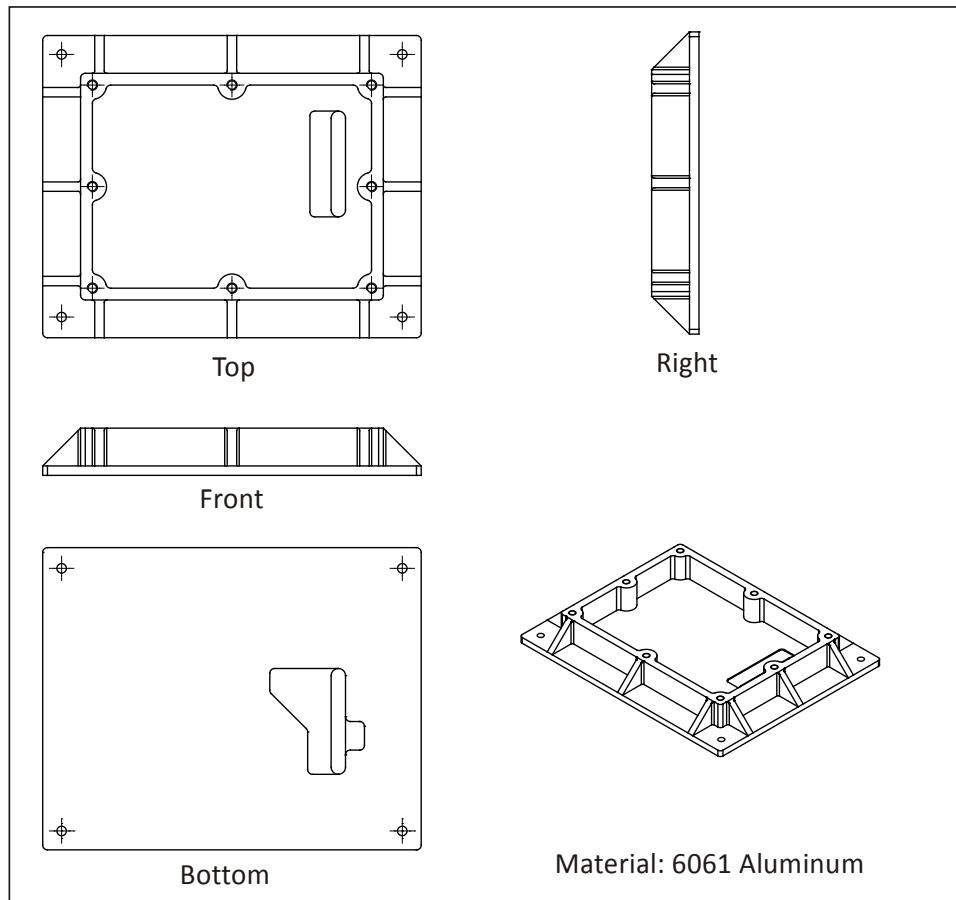


Figure 65: Microwave Housing

1 Clamp Stock to Table

Clamp the part directly to the table or sub plate. Add stock to create a flange. This is used to locate and hold the part when machining the bottom.

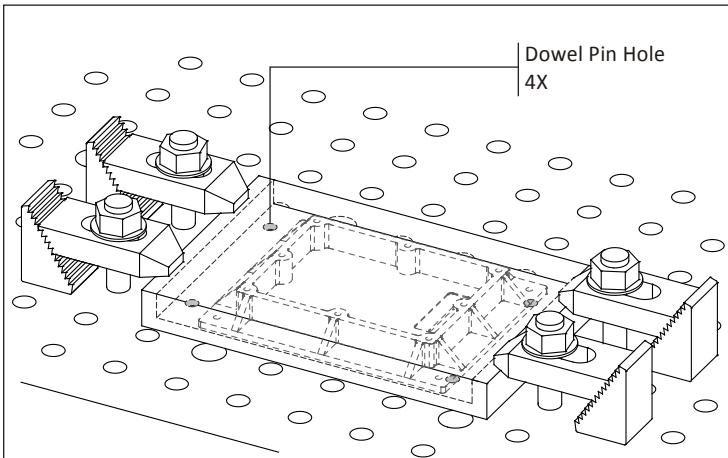


Figure 66: Microwave Housing, Top Side Setup

Figure 67 shows how the part looks after this operation, and how the dowel and shoulder bolt holes extend beyond the bottom of the part, but do not drill into the sub plate or table.

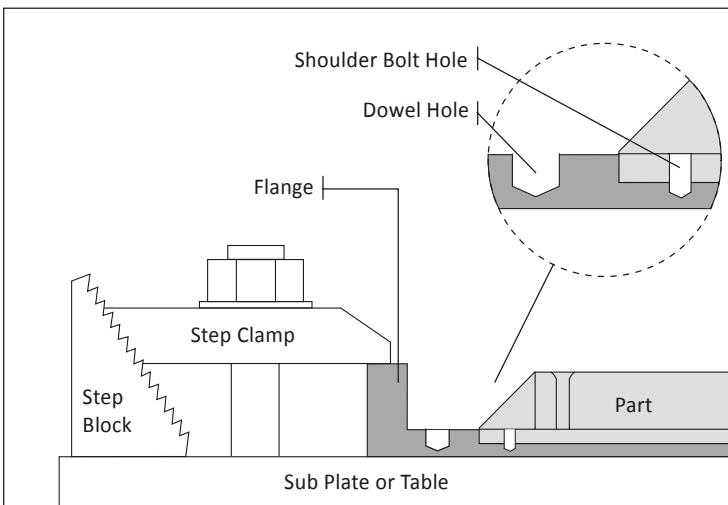


Figure 67: Front View - After Machining

2 Machine Top Complete

Machine the top complete.

Microwave Housing: Job 1	
G54	
Datum: Upper-left corner of stock.	
Op-1 Face	Face Part avoiding clamps.
Op-2 2D Rough	Rough OD avoiding clamps.
Op-3 2D Rough	Rough pocket.
Op-4 2D Finish	Finish OD. Finish the part after roughing both the inside and outside so the part will stress relieve.
Op-5 2D Finish	Finish pocket.
Op-6 3D Rough	3D rough ramps.
Op-7 3D Finish	3D finish ramps.
Op-8 2D Pocket	Rough and finish pockets in bottom of pocket.
Op-9 Ctr Drill	Center drill all holes.
Op-10 Drill	Drill holes in flange for dowel pins used to locate the part for Job 2.
Op-11 Drill	Drill holes in flange.
Op-12 Drill	Drill holes in top face of part

Figure 68: Microwave, Top Side

2 Machine Fixture Plate

Figure 69 shows a fixture plate used to machine the second side. The fixture has a cavity that the part fits into and the part is held by strap clamps on the flange.

Four dowel pin holes locate the part precisely. Notice the dowels cannot protrude above the finished thickness of the flange, so a facing tool can pass over without colliding (which would destroy the tool).

Four counter-bored holes match holes in the part. After facing, these holes are exposed, allowing shoulder bolts to be installed. These locate and clamp the part while the excess flange is cut away. Be sure to make the counterbore deep enough so the shoulder bolt will grip the part when tightened.

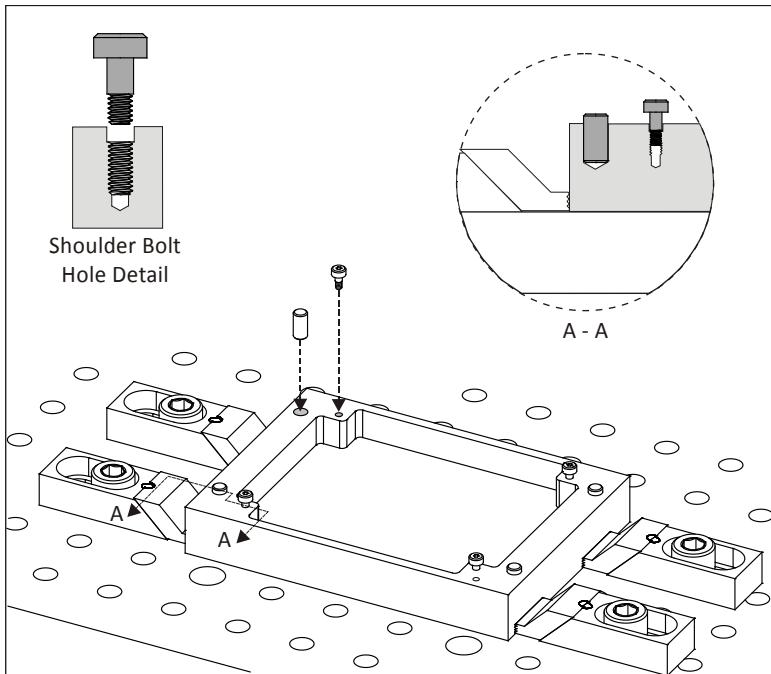


Figure 69: Fixture Plate Detail

3 Machine Fixture Plate

Machine the fixture plate given the parameters below.

Microwave Housing: Fixture Plate	
G55	
Datum: G55 XY is located at the center-face of dowel pin hole. Find this location by picking up a corner of the fixture plate stock, and then shifting G55 XY. This is important because this hole is used to precisely locate the Microwave Housing when it is fastened to this fixture.	
Op-1 Face	Face fixture plate avoiding clamps.
Op-2 2D Pocket	Rough and finish pocket.
Op-3 Ctr Drill	Center drill all holes.
Op-4 Drill	Drill holes for dowel pins.
Op-5 Drill	Drill holes for shoulder bolt threads. De-burr these holes so the chamfered edge will help guide the shoulder bolt so it slides into the hole and does not get hung on the sharp edge.
Op-6 Drill	Drill holes for shoulder bolt ground shoulder.
Op-6 Tap	Tap holes for shoulder bolts.

Figure 70: Microwave Housing, Fixture Plate

4

5 Clamp Part to Fixture

Install dowel pins and then place part face down on the fixture. Ensure the dowel pins do not extend higher than the thickness of the part flange. Make sure the part is oriented properly so the features on the bottom mate with features on the top. Use Strap clamps to hold the part by the flange. Tap the part down with a rubber mallet, and use a feeler gauge to check for gaps between the fixture and face of the part.

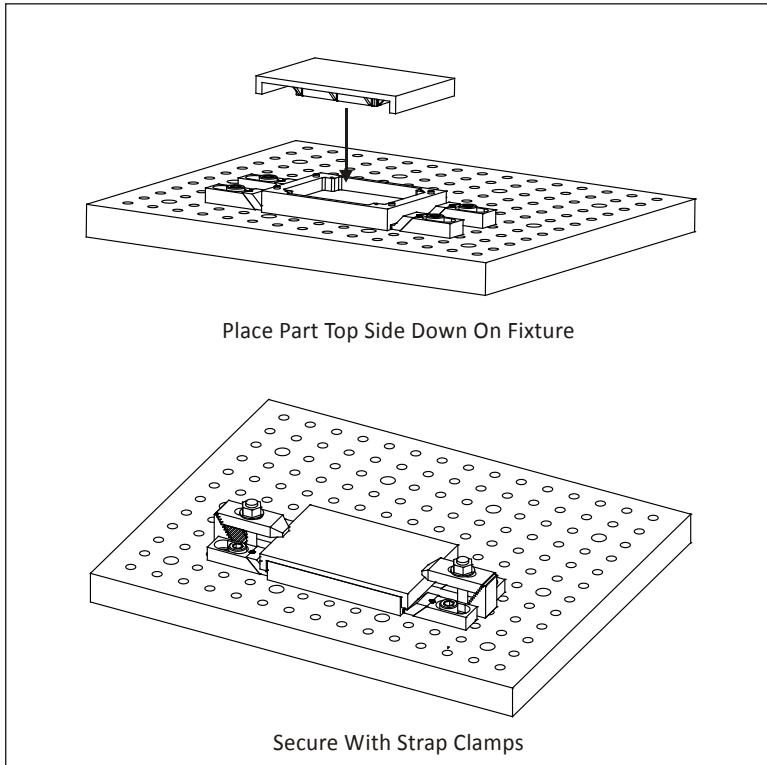


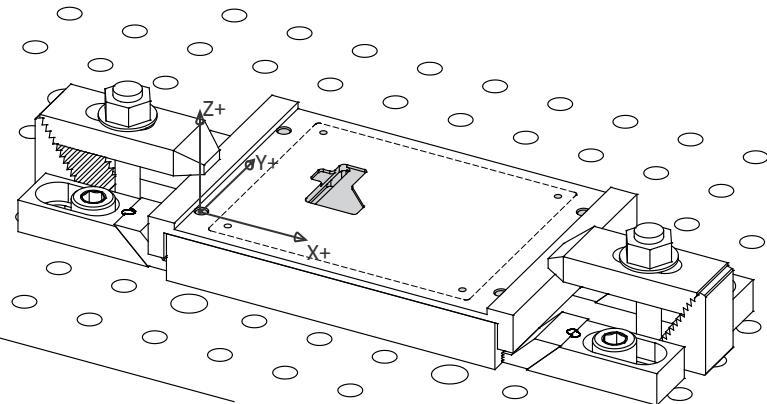
Figure 71: Locate Part on Fixture.

6 Machine Bottom Face and Cutout

Face the bottom of the part to thickness. Also machine the pocket and slot features in the bottom. Hopefully you remembered to sink the dowel pins deep enough so the tool faces over them without contacting. Notice how the dowel pins and holes for the shoulder bolts are now exposed.

Microwave Housing: Face and Machine Bottom Features

G55



Datum: Use same datum as fixture plate.

Op-1 Face	Face part.
Op-2 2D Pocket	Rough and finish pocket.

Figure 72: Face and Machine Pockets

8 Install Shoulder Bolts and Cut Away Flange

Install the shoulder bolts that are used to secure the part while the flange is cut away. You will probably have to de-burr the holes to get the bolts to slide in. Check to ensure the shoulder bolts actually grip the part. If not, then you did not make the counterbore deep enough (Step 3) and will need a washer.

Cut away the excess flange to finish the part. Leave all clamps in place during this operation to the flange will not lift during machining. Notice how close the tool passes to the dowel pins.

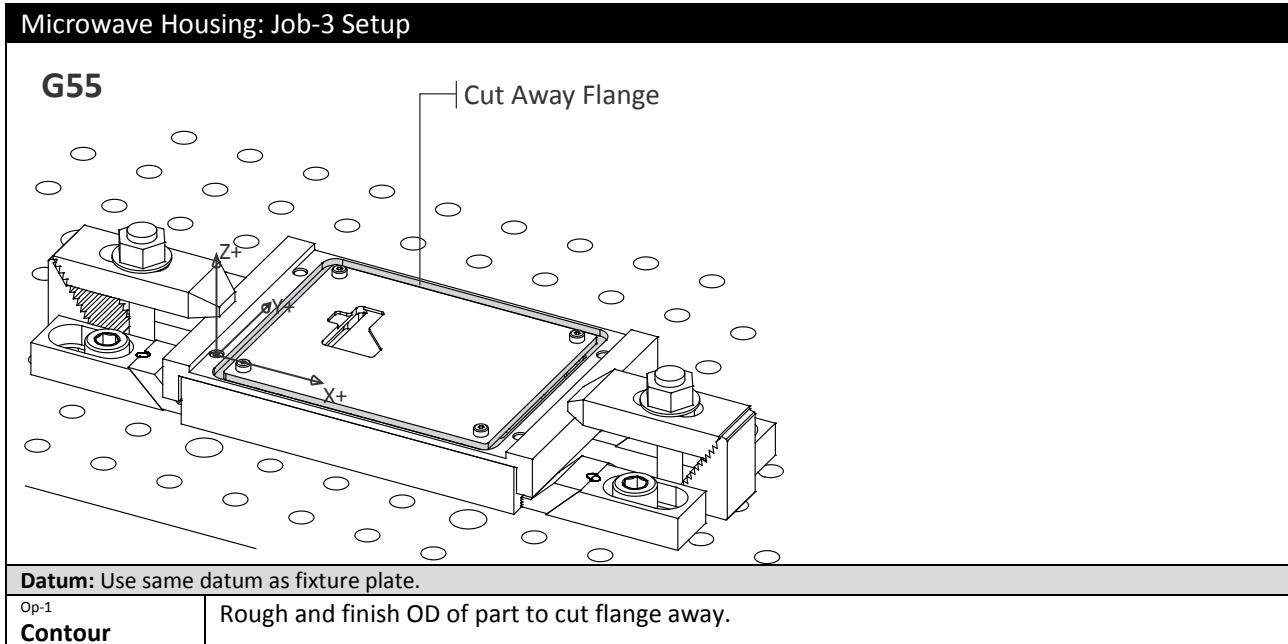


Figure 73: Cut Away Flange

Project 14: Geneva Gear

This project teaches the following skills:

- Work holding using double-sided tape.

Some parts can be held using double-sided tape. This eliminates the need for making a complex fixture and works quite well if cutting pressures are low and surface area is large enough to form a sufficient bond.

Aluminum and Medium Density Fiberboard (MDF) are commonly used fixture plate materials. Prepare an aluminum plate by clamping it to the table and facing it smooth and flat to form a good surface to bond to. Don't use coolant because its residue will inhibit good bonding.

To ensure a good bond with the tape, a good practice is to place a heavy weight on the part and let it set for at least 15 minutes after taping. The longer this weight is in place, the better the bond, so leave it for several hours or overnight if possible.

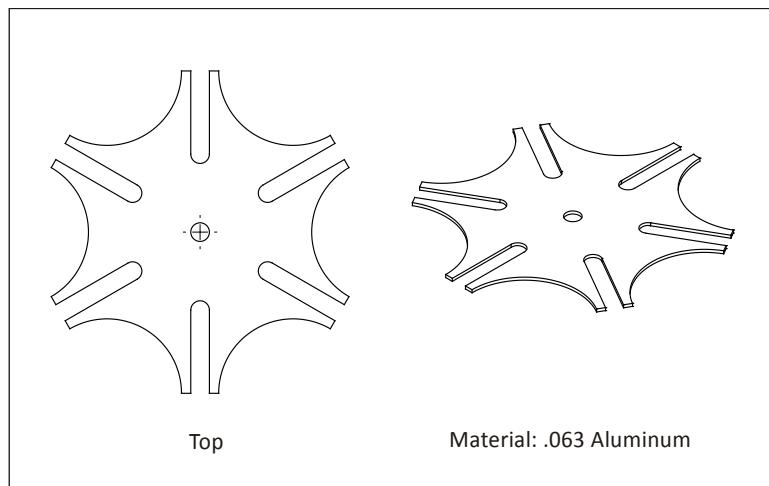


Figure 74: Geneva Gear

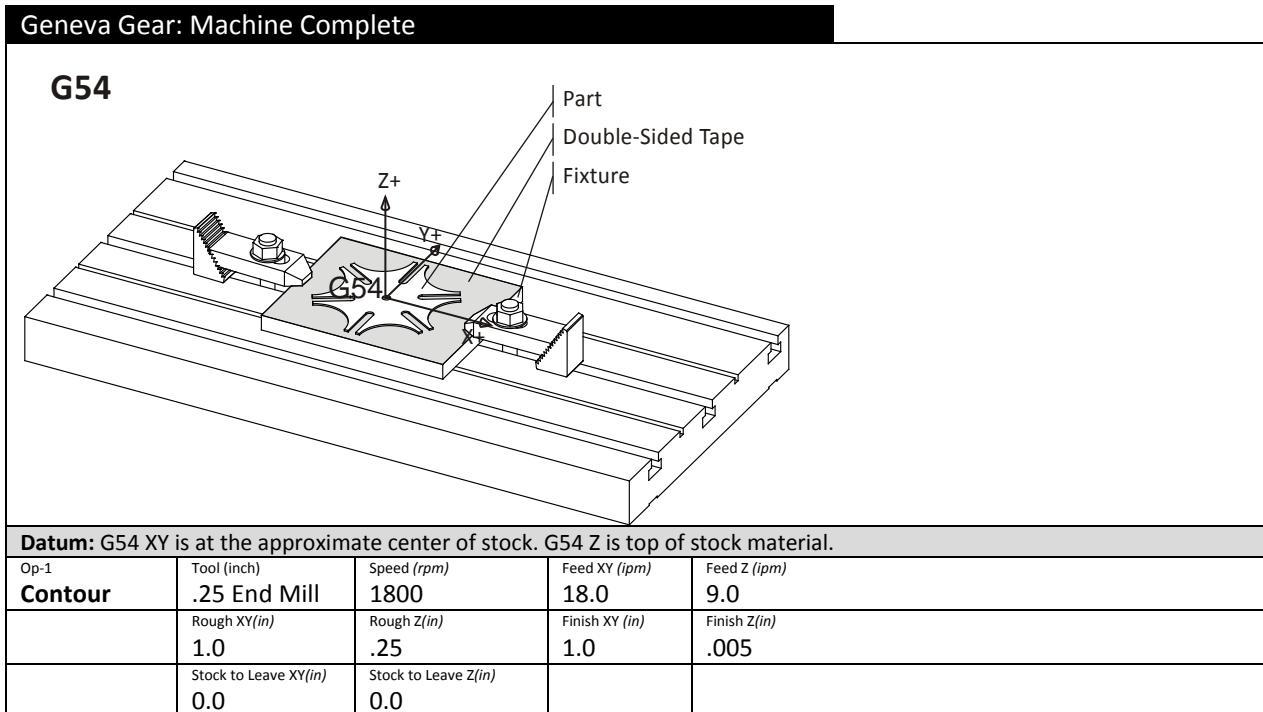


Figure 75: Geneva Gear Setup

Tips for Using Double-Sided Tape:

- Use a high quality industrial quality double-sided tape. Avoid using old tape because bonding strength decreases with shelf life.
- Avoid using coolant because it will loosen the tape.
- Weigh down the part with a heavy object to improve bonding.
- Use light cutting forces to keep the part from moving or coming completely loose during machining.
- Removing the part after machining without deforming it can be a challenge. Methods used depend on the material, but can include the use of heat or a solvent like WD-40®.

Objectives

Upon successful completion of this lesson, you will be able to:

- Set tool length offset using a scrap of notebook paper.
- Set tool length offset using a 1-2-3 Block from the finished face of the part.
- Set the tool length offset using a Tool Probe.

1.1 - Overview

One of the most confusing things about CNC's is how to set tool length offsets and how they relate to machine fixture offsets. Lessons 4 and 6 detailed one method of using Tool and Length offsets. While this is one of the most flexible and useful methods it is not easy for beginners to understand and has other disadvantages.

This chapter shows three other methods of setting tool lengths. To summarize, here is a table that lists all four methods described in this course:

Method	Description	Advantages	Disadvantages
1	Use scrap of notebook paper.	<ul style="list-style-type: none">• Easy to understand.• Easy to use.• Does not require using Fixture Offset Z.• Safe.	<ul style="list-style-type: none">• Not as accurate as other methods.• Requires part to be manually face milled before setting tools.
2	Use 1-2-3 Block on finished face of part.	<ul style="list-style-type: none">• Very accurate.• Easier to understand than method requiring use of dial indicator.	<ul style="list-style-type: none">• Not as safe as paper or tool probe method.• Requires part to be face milled before using.• Requires also setting Fixture Offset Z.• If Z-datum is milled away, tools cannot be reset easily.
3	Use 1-2-3 Block and dial indicator (Method described in Lessons 3 and 6).	<ul style="list-style-type: none">• Very accurate.• If Z-datum is milled away, tools can be reset easily in case of broken or worn tool.	<ul style="list-style-type: none">• Can be difficult to understand for beginners.• Not as safe as paper or tool probe method.• Requires also setting Fixture Offset Z.
4	Use tool probe.	<ul style="list-style-type: none">• Easy to use.• Very fast.• Accurate.• Easy to understand.	<ul style="list-style-type: none">• Requires machine equipped with tool probe.

1.2 – Using Piece of Notebook Paper

Notebook paper is remarkably consistent in thickness. This method takes advantage of this for setting tools. With this method you will place a piece of scrap paper on the finished face of the part and then jog the tool while spinning until it just contacts the paper. Contact of the tool with the paper is indicated when the paper begins spinning. The tool length offset is set to at position.

1 Face Mill Part

Begin by installing an end mill or face mill in the spindle. Follow all safety precautions. Start the spindle, and manually jogging the tool close to the top of the rough stock. Reduce the job increment to .001 inches per pulse and slowly move the tool down until it just contacts the face of the part. Then move the tool off to the side of the part and then lower the tool by the amount of the face cut (usually .005-.02inches). Increase the jog increment to .01inches per pulse and move the tool in XY as necessary to create a flat surface on the face of the part.

2 Stop the spindle

Stop the spindle before proceeding! Jog the tool well out of the way so you do not bump it with your hand or head.

3 Place Paper on Face of Part

Tear a piece of common notebook paper larger than the diameter of the tool. Place the paper so it lies flat on the face of the part.

4 Restart spindle

Follow all safety precautions to restart the spindle.

5 Move tool over scrap of paper

Manually jog the machine until the spindle is centered over the piece of paper.

6 Jog tool down until it contacts paper

Jog the tool close to the paper, and then reduce the job amount to .001 per pulse. Slowly jog the tool down .001 at a time until the tool just contacts the paper. The paper will usually begin spinning with the tool.

7 Stop the spindle

Stop the spindle but do not yet jog the machine.

8 Set the Tool Length register value

Set the Tool Length Offset register at this value. This is done on the Haas by pressing the TOOL OFFSET MESUR button (beneath F1). Other machines may require you write down the Z-value reading on the CNC control and then enter this value in the tool length register for this tool.

9 Adjust Tool Length register for paper thickness (optional)

If exact tolerances are required then the tool must be lowered by the thickness of the paper. Do this by ADDING the thickness of the paper (.003-.004 typical) to the Tool Length register.

1.3 – Use 1-2-3 Block on Finished Face of Part.

This approach is similar to the method described in Lessons 3 & 6. You must have a clean part face to begin with.

1 Face Mill Part

Begin by installing an end mill or face mill in the spindle. Follow all safety precautions. Start the spindle, and manually jogging the tool close to the top of the rough stock. Reduce the job increment to .001 inches per pulse and slowly move the tool down until it just contacts the face of the part. Then move the tool off to the side of the part and then lower the tool by the amount of the face cut (usually .005-.02inches). Increase the jog increment to .01inches per pulse and move the tool in XY as necessary to create a flat surface on the face of the part.

4 Stop the spindle

Stop the spindle before proceeding! Jog the tool well out of the way so you do not bump it with your hand or head.

5 Place 1-2-3 Block on Face of Part

Make sure there is no coolant or chips on the face of the part. Place the 1-2-3 block on the face of the part.

6 Jog Tool Alongside 1-2-3 Block

With the spindle OFF, carefully jog the tool until it is along the side of the 1-2-3 block.

7 Raise Spindle While Pressing Block Against Tool

While applying SLIGHT pressure against the side of the tool, slowly raise the spindle in .001inch increments until the tool JUST slips between the top of the part and the end of the tool. You must set the tool length based on the longest tool flute, so move the block out of the way, rotate so the next flute is pointing towards the block, and then try to slide the block underneath it. If the flute is longer than the others and the block does not slide beneath using slight pressure, raise the spindle in .001inch increments until it does. It is not uncommon for flutes to vary in length .005inches or more. Repeat until all cutting flutes have been checked.

8 Set the Tool Length Register Value

Set the Tool Length Offset register at this value. This is done on the Haas by pressing the TOOL OFFSET MESUR button (beneath F1). Other machines may require you write down the Z-value reading on the CNC control and then enter this value in the tool length register for this tool.

9 Adjust For the Length of the 1-2-3 Block

Since all tools are set 1, 2 or 3 inches (depending on which side of the 1-2-3 block used) all offsets must be lowered by this amount to adjust them downward to the top of the finished part. There are two ways to accomplish this:

- a. Adjust all tool length offset down individually.
- b. Set the Fixture Offset (Usually G54) register to the value. For example, setting G54 to 1.000 would drop the datum for all tools used 1.000 inches. This is the preferred method. It must be done AFTER setting all tool lengths.

1.4 – Use 1-2-3 Block and Dial Indicator

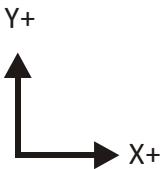
This method is covered in detail in Lessons 3 and 6. Please refer to these lessons for more information.

1.5 – Use Tool Probe

A Tool Probe is usually an optional piece of equipment on a CNC machine. It may consist of a laser (non-contact reading) or a Probe that the machine will move the tool down to touch. Tool probes allow fast, accurate, safe, and easy tool setting and are the preferred method: if your machine is equipped with a probe.

Because of the wide variety of tool probe types it is not possible to provide specific instructions in their use here. Please have your instructor or shop supervisor show you how to use the probe on your machine.

Caution: Tool and part probes are expensive. Crashing the tool or part probe can cost hundreds or thousands of dollars. Do not attempt to use these accessories without proper instruction.

CNC Process Planning Form						
Job Name:			Page: _____ of _____			
Part Number:	Rev:	Stock Size: X:	Y:	Z:		
Programmer:			Material:			
G5 _____ Fixture Offset						
			View:			
<i>Operation</i>	<i>Tool Diameter</i>	<i>Corner Radius</i>	<i>Tool Type</i>	<i>Number Flutes</i>	<i>Flute Length</i>	<i>Shoulder Length</i>
<i>Speed</i>	<i>Cutting Feed</i>	<i>Entry Feed</i>	<i>Ramp Feed</i>	<i>Plunge Feed</i>		
<i>Stepover XY</i>	<i>Finish XY</i>	<i>Stepdown Z</i>	<i>Finish Z</i>	<i>XY Stock Allowance</i>	<i>Z Stock Allowance</i>	<i>Max Depth</i>
<i>Operation Notes</i>						
<i>Operation</i>	<i>Tool Diameter</i>	<i>Corner Radius</i>	<i>Tool Type</i>	<i>Number Flutes</i>	<i>Flute Length</i>	<i>Shoulder Length</i>
<i>Speed</i>	<i>Cutting Feed</i>	<i>Entry Feed</i>	<i>Ramp Feed</i>	<i>Plunge Feed</i>		
<i>Stepover XY</i>	<i>Finish XY</i>	<i>Stepdown Z</i>	<i>Finish Z</i>	<i>XY Stock Allowance</i>	<i>Z Stock Allowance</i>	<i>Max Depth</i>
<i>Operation Notes</i>						

Page: _____ of _____

<i>Operation</i>	<i>Tool Diameter</i>	<i>Corner Radius</i>	<i>Tool Type</i>	<i>Number Flutes</i>	<i>Flute Length</i>	<i>Shoulder Length</i>
<i>Speed</i>	<i>Cutting Feed</i>	<i>Entry Feed</i>	<i>Ramp Feed</i>	<i>Plunge Feed</i>		
<i>Stepover XY</i>	<i>Finish XY</i>	<i>Stepdown Z</i>	<i>Finish Z</i>	<i>XY Stock Allowance</i>	<i>Z Stock Allowance</i>	<i>Max Depth</i>
<i>Operation Notes</i>						
<i>Operation</i>	<i>Tool Diameter</i>	<i>Corner Radius</i>	<i>Tool Type</i>	<i>Number Flutes</i>	<i>Flute Length</i>	<i>Shoulder Length</i>
<i>Speed</i>	<i>Cutting Feed</i>	<i>Entry Feed</i>	<i>Ramp Feed</i>	<i>Plunge Feed</i>		
<i>Stepover XY</i>	<i>Finish XY</i>	<i>Stepdown Z</i>	<i>Finish Z</i>	<i>XY Stock Allowance</i>	<i>Z Stock Allowance</i>	<i>Max Depth</i>
<i>Operation Notes</i>						
<i>Operation</i>	<i>Tool Diameter</i>	<i>Corner Radius</i>	<i>Tool Type</i>	<i>Number Flutes</i>	<i>Flute Length</i>	<i>Shoulder Length</i>
<i>Speed</i>	<i>Cutting Feed</i>	<i>Entry Feed</i>	<i>Ramp Feed</i>	<i>Plunge Feed</i>		
<i>Stepover XY</i>	<i>Finish XY</i>	<i>Stepdown Z</i>	<i>Finish Z</i>	<i>XY Stock Allowance</i>	<i>Z Stock Allowance</i>	<i>Max Depth</i>
<i>Operation Notes</i>						
<i>Operation</i>	<i>Tool Diameter</i>	<i>Corner Radius</i>	<i>Tool Type</i>	<i>Number Flutes</i>	<i>Flute Length</i>	<i>Shoulder Length</i>
<i>Speed</i>	<i>Cutting Feed</i>	<i>Entry Feed</i>	<i>Ramp Feed</i>	<i>Plunge Feed</i>		
<i>Stepover XY</i>	<i>Finish XY</i>	<i>Stepdown Z</i>	<i>Finish Z</i>	<i>XY Stock Allowance</i>	<i>Z Stock Allowance</i>	<i>Max Depth</i>
<i>Operation Notes</i>						

Milling G-Codes	
G00	Rapid Positioning Motion (X, Y, Z, A, B)
G01	Linear Interpolation Motion (X, Y, Z, A, B, F)
G02	Circular Interpolation Motion CW (X, Y, Z, A, I, J, K, R, F)
G03	Circular Interpolation Motion CCW (X, Y, Z, A, I, J, K, R, F)
G04	Dwell (P) (P=Seconds)
G09	Exact Stop, Non-Modal
G17	Circular Motion XY Plane Selection (G02 or G03)
G18	Circular Motion ZX Plane Selection (G02 or G03)
G19	Circular Motion YZ Plane Selection (G02 or G03)
G20	Inch Coordinate Positioning
G21	Metric Coordinate Positioning
G28	Machine Zero Return Thru Ref. Point (X, Y, Z, A, B)
G29	Move to Location Through G28 Ref. Point (X, Y, Z, A, B)
G40	Cutter Comp Cancel
G41	2D Cutter Compensation, Left (X, Y, D)
G42	2D Cutter Compensation, Right (X, Y, D)
G43	Tool Length Compensation + (H, Z)
G49	Tool Length Compensation Cancel G43/G44/G43
G52	Work Offset Positioning Coordinate
G53	Machine Positioning Coordinate, Non-Modal (X, Y, Z, A, B)
G54	Work Offset Positioning Coordinate #1
G55	Work Offset Positioning Coordinate #2
G56	Work Offset Positioning Coordinate #3
G57	Work Offset Positioning Coordinate #4
G58	Work Offset Positioning Coordinate #5
G59	Work Offset Positioning Coordinate #6
G73	HS Peck Drilling Canned Cycle (X, Y, A, B, Z, I, J, K, Q, P, R, L, F)
G74	Reverse Tapping Canned Cycle (X, Y, A, B, Z, J, R, L, F)
G76	Fine Boring Canned Cycle (X, Y, A, B, Z, I, J, P, Q, R, L, F)
G77	Black Bore Canned Cycle (X, Y, A, B, Z, I, J, Q, R, L, F)
G80	Cancel Canned Cycle
G81	Drill Canned Cycle (X, Y, A, B, Z, R, L, F)
G82	Spot Drill / Counterbore Canned Cycle (X, Y, A, B, Z, P, R, L, F)
G83	Peck Drill Deep Hole Canned Cycle (X, Y, A, B, Z, I, J, K, Q, P, R, L, F)
G84	Tapping Canned Cycle (X, Y, A, B, Z, J, R, L, F)
G85	Bore In ~ Bore Out Canned Cycle (X, Y, A, B, Z, R, L, F)
G86	Bore In ~ Stop ~ Rapid Out Canned Cycle (X, Y, A, B, Z, R, L, F)
G87	Bore In ~ Manual Retract Canned Cycle (X, Y, A, B, Z, R, L, F)
G88	Bore In ~ Dwell ~ Manual Retract Canned Cycle (X, Y, A, B, Z, P, R, L, F)
G89	Bore In ~ Dwell ~ Bore Out Canned Cycle (X, Y, A, B, Z, P, R, L, F)
G90	Absolute Positioning Command
G91	Incremental Positioning Command
G92	Global Work Coordinate System
G93	Inverse Time Feed Mode ON
G94	Inverse Time Feed OFF / Feed Per Minute ON
G98	Canned Cycle Initial Point Return
G99	Canned Cycle R Plane Return

Table 1: Mill G-Codes

Milling M-Codes	
M00	Program Stop
M01	Optional Program Stop
M02	Program End
M03	Spindle ON Clockwise (S)
M04	Spindle ON Counterclockwise (S)
M05	Spindle Stop
M06	Tool Change (T)
M08	Coolant ON
M09	Coolant OFF
M30	Program End and Reset
M31	Chip Auger Forward
M33	Chip Auger Stop
M34	Coolant Spigot Position Down, Increment
M35	Coolant Spigot Position Up, Decrement
M36	Pallet Part Ready
M41	Spindle Low Gear Override
M42	Spindle High Gear Override
M50	Execute Pallet Change
M83	Auto Air Jet ON
M84	Auto Air Jet OFF
M88	Coolant Through Spindle ON
M97	Local Sub-Program Call (P,L)
M98	Sub-Program Call (P,L)
M99	Sub-Program / Routine Return of Loop (P)

Table 2: Mill M-Codes

Note: Only one M-Code may appear in each block of code.

Lathe G-Codes	
G00	Rapid Positioning Motion
G01	Linear Interpolation Motion
G02	Circular Interpolation Motion CW
G03	Circular Interpolation Motion CCW
G04	Dwell (P) (P=Seconds)
G09	Exact Stop, Non-Modal
G18	Circular Motion ZX Plane Selection (G02 or G03)
G20	Inch Coordinate Positioning
G21	Metric Coordinate Positioning
G28	Machine Zero Return Thru Ref. Point
G29	Move to Location Through G28 Ref. Point
G32	Threading
G40	Tool Nose Compensation Cancel
G41	Tool Nose Compensation, Left
G42	Tool Nose Compensation, Right
G43	Tool Length Compensation
G49	Tool Length Compensation Cancel G43/G44/G45
G50	Spindle Speed Clamp/Set Global Coor. Offset
G51	Cancel Offset (Yasnac)
G52	Set Local Coordinate System (Fanuc)
G53	Machine Coordinate Selection
G54	Work Offset Positioning Coordinate #1
G55	Work Offset Positioning Coordinate #2
G56	Work Offset Positioning Coordinate #3
G57	Work Offset Positioning Coordinate #4
G58	Work Offset Positioning Coordinate #5
G59	Work Offset Positioning Coordinate #6
G61	Exact Stop Modal
G64	G61 Cancel
G70	Finishing Cycle
G71	OD/ID Stock Removal Cycle
G72	Face Stock Removal Cycle
G73	Irregular Path Stock Removal Cycle
G74	Face Grooving Cycle, Peck Drilling
G75	OD/ID Grooving Cycle, Peck Drilling
G76	Threading Cycle, Multiple Pass
G80	Cancel Canned Cycle
G81	Drill Canned Cycle
G82	Spot Drill / Counterbore Canned Cycle
G83	Peck Drill Deep Hole Canned Cycle
G84	Tapping Canned Cycle
G85	Bore In ~ Bore Out Canned Cycle
G86	Bore In ~ Stop ~ Rapid Out Canned Cycle
G87	Bore In ~ Manual Retract Canned Cycle
G88	Bore In ~ Dwell ~ Manual Retract Canned Cycle
G89	Bore In ~ Dwell ~ Bore Out Canned Cycle
G90	OD/ID Turning Cycle, Modal

Table 3: Lathe G-Codes

G92	Threading Cycle, Modal
G94	End Facing Cycle, Modal
G95	Subspindle Rigid Tap
G96	Constant Surface Speed (CSS) On
G97	Constant Surface Speed Cancel
G98	Feed Per Minute
G99	Feed Per Revolution

Table 3 (Cont): Lathe G-Codes

Lathe M-Codes	
M00	Program Stop
M01	Optional Program Stop
M02	Program End
M03	Spindle ON Clockwise (Forward)
M04	Spindle ON Counterclockwise (Reverse)
M05	Spindle Stop
M08	Coolant ON
M09	Coolant OFF
M10	Clamp Chuck
M11	Unclamp Chuck
M12	Auto Air Jet On
M13	Auto Air Jet Off
M14	Clamp Main Spindle
M15	Unclamp Main Spindle
M19	Orient Spindle with P value
M21	Tailstock Forward
M22	Tailstock Reverse
M23	Thread Chamfer On
M24	Thread Chamfer Off
M30	Program End and Reset
M31	Chip Auger Forward
M33	Chip Auger Stop
M36	Parts Catcher Up
M37	Parts Catcher Down
M41	Low Gear
M42	High Gear
M88	High Pressure Coolant On
M89	High Pressure Coolant Off
M133	Live Tool Drive Forward
M134	Live Tool Drive Reverse
M135	Live Tool Drive Stop

Table 4: Lathe M-Codes

Note: Only one M-Code may appear in each block of code.

Glossary

1-2-3 Block — A precision ground block that is 1x2x3 inches. Useful for helping set up the machine.

2D — A prismatic part of constant thickness.

2-1/2 D — A prismatic part with multiple thicknesses. All planes are parallel to each other.

3D — A part that is not completely prismatic, including molds and organically shaped parts.

Burr — A sharp edge or flashing.

Block — A line of code in a CNC program.

CAD — Computer Aided Design

CAM — Computer Aided Manufacturing.

CDC — Cutter Diameter Compensation.

CNC — Computer Numerical Control.

Chip Load — The amount of material removed with each pass of a tool cutting edge.

Datum — Point on the part from which all coordinates are referenced.

Drill Chart — A table that shows drill sizes, their decimal equivalent, and what size drill to use for a specified tap.

Feed — The linear speed at which a tool advances through material.

First Article — The first part machined with a new program and setup.

G-M Code — The CNC machine tool language.

Gouge — An error that causes an overcut on the part.

High Speed Machining (HSM) — Toolpath strategies that minimize machining time and tool breaking. Includes several technologies such as Adaptive Roughing that maintain a constant amount of tool engagement.

Home — Location of machine after it is turned on and moved to its start position.

ID — Inside diameter or contour, like a pocket.

Insert — Carbide composite cutting tool used extensively for turning operations.

Job — A sequential list of one or more machining operations that share the same setup and fixture offset.

Machine Coordinate System — Coordinates in reference to the machine Home position.

Mill — A machine tool that removes material by spinning a tool and moving it in relation to the part.

OD — Outside diameter or contour.

Oversize — Describes a machined feature is too big because not enough material was removed during machining.

Post Processor — The part of a CAD/CAM system that actually creates and formats the CNC program for a specific machine and control.

Rapid — The fastest linear feed rate at which a machine tool can move.

Reference Point — Some point on the part that can be found reliably by mechanical means.

Rigid Tapping — The capability of a CNC machine to control a tap without the use of a special tapping attachment.

Scallop — Ridges left on the part, usually from a ball or bull nose end mill on a 3D sculpted surface.

Setup — A complete set of work holding to machine one side of a part.

Speed — How fast a tool turns in RPM.

Spindle — Part of CNC machine that grips and spins the cutting tool.

Stepdown — Distance the tool moves down in Z between machining passes.

Stepover — Distance the tool moves over in the XY plane between machining passes.

Stock Allowance — Amount of material left after a machining operation or pass to be removed by a finish pass.

Spring Pass — Two or more machining passes that follows the exact same path. Used to straighten walls and improve surface finish.

TLO — Tool Length Offset. Used to tell CNC machine the length of a tool.

Tool Changer — A device on a CNC machine that automatically takes tools from a carousel and places them in the machine spindle.

T-Slot — A slot in the machine table into which a special T-nut can be inserted. Used to fasten clamps or parts to the machine table.

Turret — Lathe assembly that attaches the lathe tool holder to the machine.

Undercut — Describes a machined feature is too small because too much material was removed during machining.

Decimal Equivalent Chart							
Decimal Equivalent	Drill Size	mm	Tap Size	Decimal Equivalent	Drill Size	mm	Tap Size
.0059	97	0.150		.0330	66	0.838	
.0063	96	0.160		.0350	65	0.889	
.0067	95	0.170		.0360	64	0.914	
.0071	94	0.180		.0370	63	0.940	
.0075	96	0.191		.0380	62	0.965	
.0079	92	0.201		.0390	61	0.991	
.0083	91	0.211		.0400	60	1.016	
.0087	90	0.221		.0410	59	1.041	
.0091	89	0.231		.0420	58	1.067	
.0095	88	0.241		.0430	57	1.092	
.0100	87	0.254		.0465	56	1.181	
.0105	86	0.267		.0469	3/64	1.191	#0-80
.0110	85	0.279		.0520	55	1.321	
.0115	84	0.292		.0550	54	1.397	
.0120	83	0.305		.0595	53	1.511	#1-64 #1-72
.0125	82	0.318		.0625	1/16	1.588	
.0130	81	0.330		.0635	52	1.613	
.0135	80	0.343		.0670	51	1.702	
.0145	79	0.368		.0700	50	1.778	#2-56 #2-64
.0156	1/64	0.397		.0730	49	1.854	
.0160	78	0.406		.0760	48	1.930	
.0180	77	0.457		.0781	5/64	1.984	
.0200	76	0.508		.0785	47	1.994	#3-48
.0210	75	0.533		.0810	46	2.057	
.0225	74	0.572		.0820	45	2.083	#3-56
.0240	73	0.610		.0860	44	2.184	
.0250	72	0.635		.0890	43	2.261	#4-40
.0260	71	0.660		.0935	42	2.375	#4-48
.0280	70	0.711		.0938	3/32	2.381	
.0292	69	0.742		.0960	41	2.438	
.0310	68	0.787		.0980	40	2.489	
.0313	1/32	0.794		.0995	39	2.527	
.0320	67	0.813		.1015	38	2.578	#5-40

Table 1: Inch Drill and Tap Sizes

Decimal Equivalent Chart							
Decimal Equivalent	Drill Size	Mm	Tap Size	Decimal Equivalent	Drill Size	mm	Tap Size
.1040	37	2.642	#5-44	.1960	9	4.978	
.1065	36	2.705	#6-32	.1990	8	5.055	
.1094	7/64	2.778		.2010	7	5.105	1/4-20
.1100	35	2.794		.2031	13/64	4.159	
.1110	34	2.819		.2040	6	5.182	
.1130	33	2.870	#6-40	.2055	5	5.220	
.1160	32	2.946		.2090	4	5.309	
.1200	31	3.048		.2130	3	5.410	1/4-28
.1250	1/8	3.175		.2188	7/32	5.556	
.1285	30	3.264		.2210	2	5.613	
.1360	29	3.454	#8-32 #8-36	.2280	1	5.791	
.1405	28	3.569		.2340	A	5.944	
.1406	9/64	3.572		.2344	15/64	5.953	
.1440	27	3.658		.2380	B	6.045	
.1470	26	3.734		.2420	C	6.147	
.1495	25	3.797		.2460	D	6.248	
.1520	24	3.861		.2500	1/4 & E	6.350	
.1540	23	3.912		.2570	F	6.528	5/16-18
.1563	5/32	3.969		.2610	G	6.629	
.1570	22	3.988		.2656	17/64	6.747	
.1590	21	4.039	#10-32	.2660	H	6.756	
.1610	20	4.089		.2720	I	6.909	5/16-24
.1660	19	4.216		.2770	J	7.036	
.1695	18	4.305		.2810	K	7.137	
.1719	11/64	4.366		.2813	9/32	7.144	5/16-32
.1730	17	4.394		.2900	L	7.366	
.1770	16	4.496		.2950	M	7.493	
.1800	15	4.572		.2969	19/64	7.541	
.1820	14	4.623		.3020	N	7.671	
.1850	13	4.699		.3125	5/16	7.938	3/8-16
.1875	3/16	4.763	#12-32	.3160	O	8.026	
.1890	12	4.801		.3230	P	8.204	
.1910	11	4.851		.3281	21/64	8.334	
.1935	10	4.915		.3320	Q	8.433	3/8-24

Table 1: Inch Drill and Tap Sizes

Decimal Equivalent Chart							
Decimal Equivalent	Drill Size	mm	Tap Size	Decimal Equivalent	Drill Size	mm	Tap Size
.3390	R	8.611		.6094	39/64	15.478	11/16-12
.3438	11/32	8.731	3/8-32	.6250	5/8	15.875	
.3480	S	8.839		.6406	41/64	16.272	11/16-20 11/16-24
.3580	T	9.093		.6563	31/32	16.669	¾-10
.3594	23/64	9.128		.6719	41/64	17.066	
.3680	U	9.347	7/16-14	.6875	11/16	17.462	¾-16
.3750	3/8	9.525		.7031	45/64	17.859	¾-20
.3770	V	9.576		.7188	23/32	18.256	
.3860	W	9.804		.7344	47/64	18.653	13/16-12
.3906	25/64	9.922	7/16-20	.7500	¾	19.050	13/16-16
.3970	X	10.084		.7656	49/64	19.447	13/16-20 7/8-9
.4040	Y	10.262	7/16-28	.7813	25/32	19.844	
.4063	13/32	10.319		.7969	51/64	20.241	7/8-14
.4130	Z	10.490		.8125	13/16	20.637	
.4219	27/64	10.716	½-13	.8281	53/64	21.034	7/8-20
.4375	7/16	11.113		.8138	27/32	21.431	
.4531	29/64	11.509	½-20	.8594	55/64	21.828	15/16-12
.4688	15/32	11.906	½-28	.8750	7/8	22.225	15/16-16 1.0-8
.4844	31/64	12.303	9/16-12	.8906	57/64	22.622	15/16-20
.5000	½	12.700	9/16-18	.9063	29/32	23.019	
.5156	33/64	13.097	9/16-24	.9219	59/64	23.416	1.0-12
.5313	17/32	13.494	5/8-11	.9375	15/16	23.813	
.5469	35/64	13.891		.9531	61/64	24.209	1.0-20
.5625	9/16	14.288	5/8-18	.9688	31/32	24.606	
.5781	37/64	14.684	5/8-24	.9844	63/64	25.003	
.5938	19/32	15.081		1.000	1	25.400	

Table 1: Inch Drill and Tap Sizes

Inch Pipe Thread Taps			
Tap Thread Size	Approximate Inside Diameter	Approximate Outside Diameter	Tap Drill
1/8-27	1/4	3/8	11/32
1/4-18	3/8	17/32	7/16
3/8-18	1/2	11/16	37/64
1/2-14	5/8	13/16	23/32
5/8-14	13/16	1	59/64
1-11 1/2	1 1/16	1 5/16	1 - 5/32
1 1/4-11 1/2	1 3/8	1 5/8	1 1/2
1 1/2-11 1/2	1 5/8	1 7/8	1 - 47/64
2-11 1/2	2 1/16	2 3/8	2 - 7/32
2 1/2-8	2 9/16	2 7/8	2 - 5/8

Table 2: Pipe Thread Taps

Metric Taps					
Decimal Equivalent (Inches)	Drill Size (mm)	Tap Size	Decimal Equivalent (Inches)	Drill Size (mm)	Tap Size
.0492	1.25	M1.6X0.35	.3937	10	M12X1.75
.0571	1.45	M1.8X0.35	.4134	10.5	M12X1.25
.0630	1.6	M2X0.4	.4724	12	M14X2
.0689	1.75	M2.2X0.45	.4921	12.5	M14X1.5
.0748	1.9	M2.3X0.4	.5118	13	M14X1.25
.0807	2.05	M2.5X0.45	.5512	14	M16X2
.0846	2.15	M2.6X0.45	.5709	14.5	M16X1.5
.0984	2.5	M3X0.5	.6102	15.5	M18X2.5
.1142	2.9	M3.5X0.6	.6299	16	M18X2.0
.1299	3.3	M4X0.7	.6890	17.5	M20X2.5
.1476	3.75	M4.5X0.75	.7087	18	M20X2.0
.1654	4.2	M5X0.8	.7283	18.5	M20X1.5
.1969	5	M6X1	.7677	19.5	M22X2.5
.2067	5.25	M6X0.75	.7874	20	M22X2.0
.2362	6	M7X1	.8071	20.5	M22X1.5
.2657	6.75	M8X1.25	.8268	21	M24X3
.2756	7	M8X1.00	.8661	22	M24X2
.3051	7.75	M9X1.25	.8858	22.5	M24X1.5
.3346	8.5	M10X1.5	.9449	24	M27X3
.3445	8.75	M10X1.25	.9843	25	M27X2
.3543	9	M10X1.00	1.0433	26.5	M30X3.5
.3740	9.5	M11X1.50	1.024	28	M30X2

Table 3: Metric Taps

