Hi and welcome to my free online

MACHINE SHOP COURSE FOR NOVICE MACHINISTS

As you work your way through this document, you will encounter certain titles that appear in blue. These are links to pertinent videos that I have produced. The idea here is to read the notes and watch the corresponding videos as they are suggested, I believe that following the proper order will increase your learning since you may find yourself spending time on less glamourous subjects that are all too often overlooked such as safety, mathematics, materials, blueprint reading .... This is a work in progress and I will add to it as I produce new videos so check my web page regularly.

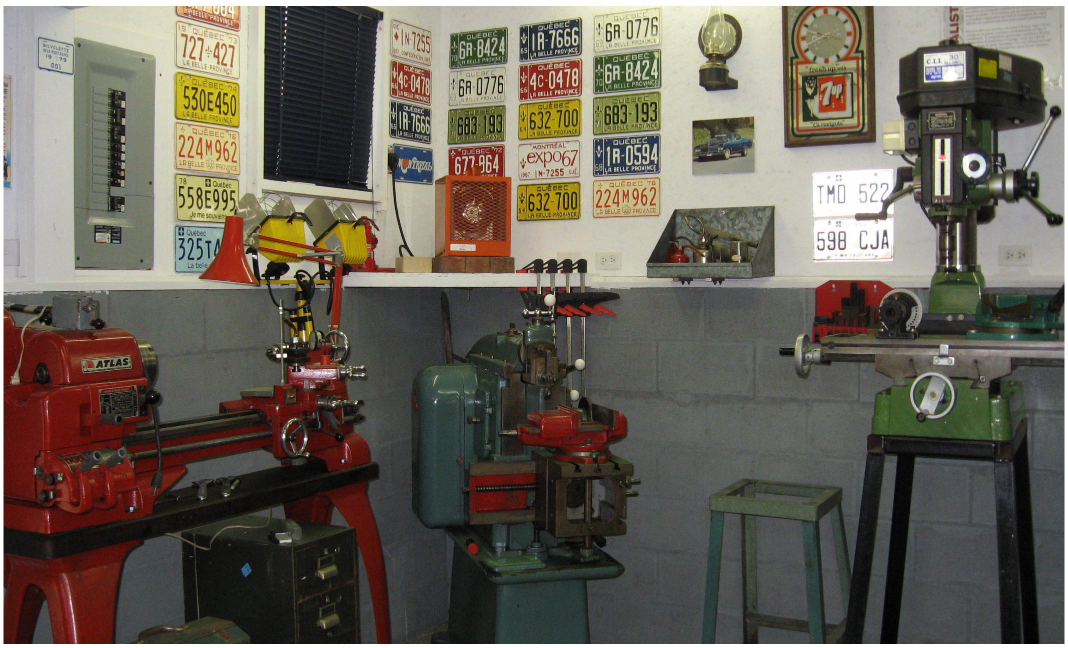
This is version 03 November 2020

INDEX:

SECTION 1: INTRODUCTION

ACCOMPANYING VIDEO FOR THIS SECTION:

MACHINE SHOP LESSON 1, INTRODUCTION



You can find all of my videos at
THATLAZYMACHINIST.COM

1-A: DEFINITIONS

Machining:

To make or finish a part that must be dimensionally and geometrically accurate by material removal using machine tools. It is an accurate sculpting operation that produces useful parts.

Machine tool:

Group of machines, that incorporate at least one accurate linear axis of motion, that are designed to remove a controlled amount of material from a part (either by chip productions, by material fusion or by material vaporization) generally referred to as machining.

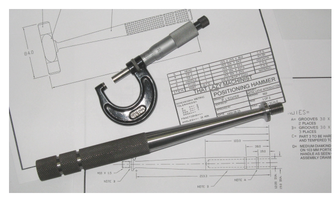
There are several types of machine tools, the most common are: the drill press, the engine lathe, the milling machine (vertical, horizontal) and the surface grinder.

Machine shop:

A space in which is installed a number of machine tools, bench work tools, measuring tools and heat treatment equipment. Type, size and quality of equipment vary with type of work performed, there are no good or bad shops! Bigger is not better! The only shop to be avoided is the unsafe shop!

Precision:

In the machine shop, precision can be defined in two ways: The amount of acceptable error that a part can incorporate (tolerance) or the smallest amount of material that can be removed on a part for a specific operation (a surface grinder is more accurate than a mill because it can remove less material per pass).



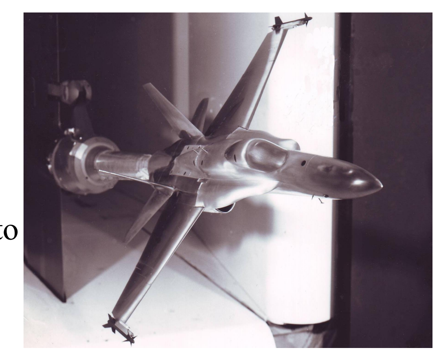
1-B: WHAT JUSTIFIES MACHINING?

Machining is by far the most expensive way to mass produce mechanical parts. However, when a small number of parts are required, when the operations cannot be completed any other way or when the parts precision requires it, machining remains the best way to go.

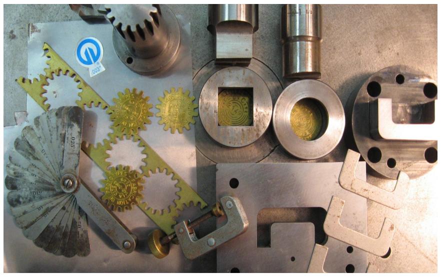
Machining is generally used for **four types of industrial activities**: Research and development, mass production, finishing and repair (maintenance) work.

#1: RESEARCH AND DEVELOPMENT

Prototype development and instrument making require machining since many of the parts produced are one off parts. Since very few of these parts are produced it is not cost effective to produce productions tooling.



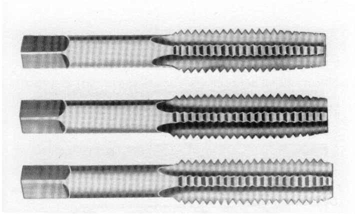
#2: **MASS PRODUCTION (MANUFACTURING)**



Although used less and less, manual machining can still hold its own for short runs on simple parts requiring few different operations. However CNC machining is widely used in short and medium runs on parts that require multiple operations.

It is important to remember that mass producing parts requires specialized tooling (moulds, punch and dies, fixtures ...) that need to be produced by very competent machinist, toolmakers, mould makers ... These specialized tools are often produced in tool rooms using traditional or CNC machines.

#3: FINISHING OPERATIONS (SECONDARY OPERATIONS)



Finishing operations include all the machining operations required to finish a part that has been previously shaped by another type of operation (moulding,, forging ...). These semi-finished parts often require machining to increase precision and to incorporate such things as threaded holes. An engine block is a good example of this.

#4: REPAIR AND MAINTENANCE WORK

Some parts just can't be bought, and when that's the case, they are machined!

<u>1-C: ATTITUDES AND APTITUDES</u>

In the machine shop the part to produce is invariable. It is only acceptable if perfect. This implies that the machinist's talent and creativity have nothing to do with the end result, but they have everything to do with the road that leads to that perfect finished product.

The challenge is making a perfect part, the pleasure is the road that you choose to get there!

Long term planning, a sequential approach to problem solving, an innovative mind and good spatial perception are the hallmarks of a good machinist.

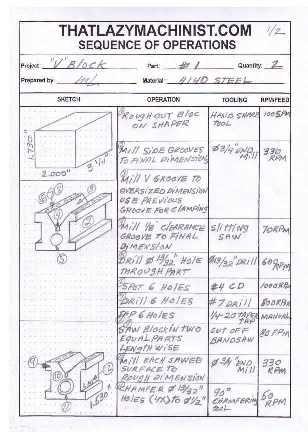
<u>1-D: WHAT YOU NEED TO KNOW TO SUCCEED!</u>

In order to succeed in the machine shop, one must become competent in these four areas:

#1: MACHINING

With a minimum of training, anyone can operate a machine tool. The challenge isn't operating a machine tool, the challenge is being creative with machine tools. A machine operator is concerned with one or two operations to perform on a large number of parts. The operator often receives specific instructions concerning only the operations that he or she will perform. This is good honest work, and quite often it is the starting point for apprentice machinists. To advance ones career, a machinist or toolmaker must learn to "see" the whole project and to understand the entire sequence of operations. A good machinist knows how to operate many different types of machine tools and plan a sequence of operations that will lead to that perfect part. It's the difference between playing the violin and being a virtuoso.

#2: SKETCHING AND BLUEPRINT READING



THATLAZYMACHINIST.COM 1/2
SEQUENCE OF OPERATIONS

Project: "V" BLOCK Part: #1 Quantity: 2
Prepared by: ML Material: 4140 STEEL

SKETCH	OPERATION	TOOLING	RPM/FEED
	ROUGH OUT BLOC ON SHAPER	HAND SHARP TOOL	100SPM
	MILL SIDE GROOVES TO FINAL DIMENSION	Ø3/4" END MILL	330 RPM
	MILL V GROOVE TO OVERSIZED DIMENSION USE PREVIOUS GROOVE FOR CLAMPING		
	MILL 1/8" CLEARANCE GROOVE TO FINAL DIMENSION	SLITTING SAW	70RPM
	DRILL Ø 13/32" HOLE THROUGH PART	Ø13/32" DRILL	600 RPM
	SPOT 6 HOLES	#4 CD	1000RPM
	DRILL 6 HOLES	#7 DRILL	800RPM
	TAP 6 HOLES	1/4-20 TAPER TAP	MANUAL
	SAW BLOCK IN TWO EQUAL PARTS LENGTHWISE	CUT OFF BANDSAW	80 FPM
	MILL EACH SAWED SURFACE TO ROUGH DIMENSION	Ø 3/4" END MILL	330 RPM
	CHAMFER Ø 13/32" HOLES (4X) TO Ø1/2"	90° CHAMFERING TOOL	50 RPM

A machinist must be able to sketch and must know how to read complex blueprints. Sketching and drawing are a machinist's way of clearly communicating with the often uninitiated client. Clients and bosses will be very happy if problems in the design can be found by a machinist (who has mastered blueprint reading) before the part is made. Removing material is a lot easier than adding it.

#3: SCIENCES

In order to perform properly in the shop, a machinist must have a good basic understanding of mathematics (algebra and trigonometry), physics (what happens to parts when force is applied on them) and materials. The sciences that most of us studied in high school were quite often very theoretical. The problems a machinist faces everyday that require this knowledge to resolve are real. The math and science that we need is basic, but must be resolved accurately and perfectly. It is as important as cutting, as far as getting the perfect part is concerned. Everything that a machinist calculates is checked and rechecked because the boss isn't a math teacher and almost right is no good.

#4: CREATIVITY

No one can be creative in the shop without knowledge. A person who is creative in the shop without knowledge is called lucky (lucky that you haven't killed yourself yet!).

Take the time to accumulate knowledge and experience. This course and the accompanying practical videos will help you down that road. Take time to learn, don't start by making crucial parts for your helicopter, because if something goes wrong the consequences could be dire. Start by making new bushings for your push lawnmower wheel, take the time to get a feel for things and gradually work your way up. Creativity and confidence will come with time.

1-E: A FEW SUGGESTIONS!

#1: In the machine shop, haste must never replace preparation.

#2: The dimensions and tolerances on a blueprint are the only ones that matter.

#3: It is impossible to produce accurate parts without accurate measurements.

#4: The sequence of operations is crucial and must be planned out before cutting.

#5: Precision is quite often the result of successive operations. The more a part needs to be precise, the more operations will be required to complete it.

#6: Every machining operation has the potential of destroying your part. The sequence of operations must be planned in a way that maximizes your chances of success.

SECTION 2: SAFETY

ACCOMPANYING VIDEOS FOR THIS SECTION:

MACHINE SHOP LESSON 2, SAFETY PART 1

also

MACHINE SHOP LESSON 2, SAFETY PART 2

also

MACHINE SHOP LESSON 2, SAFETY PART 3

also

MACHINE SHOP LESSON 2, SAFETY PART 4

2-A: INTRODUCTION

Today's society provides us with a protective cocoon that, even if designed for our safety, can sometimes help us lower our guard and become vulnerable to accidents.

Over protection can sometimes have perverse effects.

In the seventies, minor hockey associations introduced new rules in order to reduce the large numbers of facial injuries. It was logical to suppose that the introduction of a mandatory face shield would reduce facial injuries. And it did!

Sadly though, the reduction in facial injuries was accompanied by an increase in neck injuries. The hockey associations didn't foresee that the increased protection would also, since the players were less afraid of injury, increase the player's sense of invulnerability and push them to play more aggressively. People who do not feel fear when faced with a dangerous situation are a menace to themselves and to others. Fear increases our level of vigilance and prepares us to react quickly when something goes wrong. I think that it's safe to say that

NO FEAR = NO BRAINS !

A machine shop is not by definition a safe place to be. If used improperly, all machine tools can injure and maim, some can even kill! If this does not inspire you to work safely and to be attentive in the shop, well perhaps you have chosen the wrong trade. Even though I have worked in shops for years and years, I still remind myself each time I approach a machine that I have ten fingers and two eyes and that I want to finish the job with all my body parts intact.

2-B: IMPORTANT WORK HABITS TO ADOPT

#1: We all try to catch things (with our hands or feet) that fall accidentally. In the shop we have to get used to letting things fall and fixing the problems later. If we get used to catching little things (pencils, glasses ...), it becomes probable that we will try to catch larger or much sharper things (vices, steel bars, milling cutters ...). A habit quickly becomes a reflex.

#2: Many of us have the habit of trying to stop (with our hands) a moving mechanical assembly when something goes wrong. A classic example would be trying to stop by hand a spinning lathe chuck that is becoming loose on the lathe spindle. When something goes very wrong, here is a good way to control your instincts:

2.1: Take a step backwards placing your hands to each side of your body.

2.2: Take a deep breath and locate the nearest emergency stop button.

2.3: Activate the emergency stop and walk away!!!

#3: It is important to develop the habit of never adjusting a machine tool (installing parts, installing tools & accessories ...) that is not turned off by at least two switches. If you are repairing a machine its electrical feed must be locked out.

#4: Avoid quick movements, get used to working at a safe and steady pace.

#5: Work standing up and avoid leaning on machine tools.

#6: Keep your work area clean, nothing on the ground to trip or slip on! No piles of tools on machines or benches that can fall!

#7: Be aware of what is going on around you. You may be working safely, but is your neighbour doing the same? You could be injured by someone else.

#8: Read up on, study and learn about a machine tool before turning it on!

#9: Don't suffer in silence! If something goes wrong make some noise! Others can help. If you are working alone have an intercom system on at all times. An otherwise moderate injury can become disastrous if no one is around to assist.

#10: Never, never, never ever leave a chuck key in a chuck! Even for one second.

#11: Get used to holding chuck keys with the same hand that you use to start machines.

#12: Wait for a machine to stop moving completely before adjusting anything, before cleaning anything or before removing anything.

#13: A machine tool must be adjusted and operated by one person and only one person.

#14: Never, never, never ever let your fingers (if you wish to keep them, other than in a box) get close to a moving or rotating cutter!

#15: Never let an injured person drive or seek medical attention alone!

#16: Always wear eye protection in the shop!

#17: Steel toe work boots or shoes preferably with anti skid soles must be worn at all time in the shop.

#18: Clothing in the shop should conform to these requirements:

18.1: Material should be easily torn and fire resistant, cotton is preferred.

18.2: Loose fitting clothes that can become entangled in machine parts must be avoided.

18.3: Long sleeves should be rolled above elbow, short sleeves are preferred.

18.4: Pants must fall over top of boots or shoes, no cuffs and no shorts allowed.

#19: Rings, watches, bracelets, neck chains, neck ties or any other accessories that can cause or aggravate an accident are to be avoided.

#20: Long hair (more than two inches) must be controlled in a cap or bonnet. Pony tails and braids are not acceptable.

#21: As far as alcohol or prescription drugs go, if the law or your doctor says that you should not be driving, you should not be working in a shop.

#22: The shop should be a fun place to be, but it is important to always remember that it is a dangerous place to be, so roughhousing and pranks are not welcome.

#23: It is recommended to never work alone in the shop, for home shops, an intercom and letting someone who can check on you know that you will be working in the shop can help keep things safe...

#24: Always follow the safety recommendations and instructions provided with your equipment. If you do not have these documents, contact your manufacturer to get a copy.

#25: Never render inoperative the safety features of your equipment. Manufacturers go out of their way to make safe machines, when they incorporate a feature it is because it is required!

#26: **THINK FIRE SAFETY!!!!** have your shop inspected by your local fire department.

2-C: MOVING HEAVY OBJECTS

Lifting and moving heavy objects is responsible of many shop injuries. It is not necessary to be incredibly strong to move heavy objects safely; common sense and following these safety rules will get your parts where they need to be.

#1: **Be conservative and realistic when evaluating an objects weight**. Always suppose that it will be heavier than you think.

#2: **Plan your move!** Make sure that the path that joins where you are to where you have to be is clear and clean. Prepare the objects final resting place before you lift!

#3: **Pain can change an objects weight!** Even a relatively light object can become "heavy" if it is difficult to hold. Things that have odd shapes, things that don't have good gripping surfaces (very profiled parts) and things that have sharp edges can surprise you since they are easy to lift but difficult to hold!

#4: **Lift with your legs not with your back!** Your back must remain straight and vertical when lifting.

#5: **Never twist your back**! When turning during a lift it is crucial to turn with your feet avoiding twisting your spine.

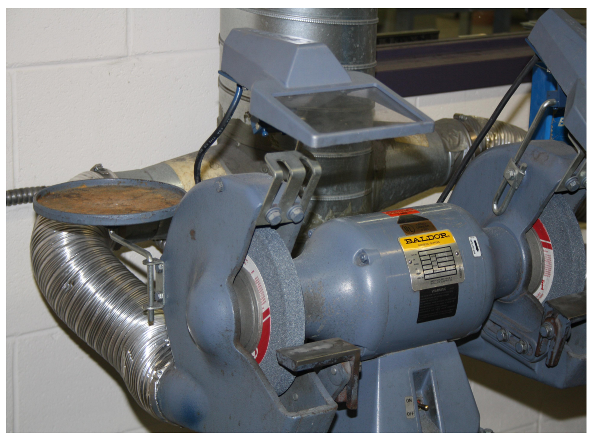
#6: **Trust your brain more than your brawn!** If you have any doubt about a lift, use the appropriate equipment. Hydraulic tables, roller cranes, floor jacks, overhead cranes, "A" frames and come along etc. are there to help and are just as important as lathes and mills in a shop.

#7: **Lifting heavy objects should be a solo job**, communication problems can make lifting in teams very dangerous. I generally recommend that you use equipment rather than buddies. You can damage equipment, but you can't kill it.

2-D: SOME SPECIFIC DANGERS

In order to reduce immediately the chance of accidents, let's look at a few safety rules that apply specifically to certain machines.

BENCH GRINDER



#1: The tool rests, the side guards, the spark guards and the face shield are integral parts of your bench grinder, they must be functional and adjusted regularly.

#2: The grinding wheel you use must be certified for your grinders R.P.M.

#3: Avoid grinding non ferrous metals.

#4: Use the tool rest to stabilize your work part and maintain a safe distance between your fingers and the wheel.

NOW WOULD BE A GOOD TIME TO WATCH FROM A SAFETY PERSPECTIVE:

INTRODUCTION TO THE BENCH GRINDER, GRINDING 101

DRILL PRESS

#1: Long hair can get entangled in the spindle. This may sound funny but it kills and disfigures many people every year.

#2: We all know that we must never leave a key in a chuck (drill press or lathe). Less people know that it is dangerous to attach a drill chuck key to a machine using a chain or a cord. It can make a bad accident worst, really quickly.

#3: If your drill press is not equipped with a rack and pinion (rack and pinion gears being self locking) for lifting or lowering the table, use a clamp on the column to avoid dropping the table all the way to the base (crushing foot injuries).

#4: Never use a milling vice on the drill press (other than a radial arm drill press). Drill press vices are made to be light, easy to handle and stable even when not clamped to a table.

#5: Thin parts, soft parts and large parts can all be snagged by a drill bit. What can you do?

Thin parts: Sandwich thin parts between thicker parts of a similar material or ***assemble a support drill jig*** directly on the work table.

Soft parts: Brass, copper, lead and most plastics can get snagged by the drill bit at any moment. ***Straight flute drills or helical drills*** with small flat produced on the cutting edges must be used for these materials.

Large parts: Large parts that are not fixed to the table must be ***blocked by a fixed pin*** in order to eliminate any possibility of rotation.

#6: Chips and debris left on a drill press table as well as poorly deburred holes can destabilize parts and cause accidents.

#7: A drill press is not a milling machine or a router.

NOW WOULD BE A GOOD TIME TO WATCH FROM A SAFETY PERSPECTIVE:

THE DRILL PRESS PART 1

also

1-2-3 BLOCK PROJECT PART 2

also

DRILL POINT GAUGE PART 1

BAND SAW

#1: Adjust the height of the blade guard for each part sawed.

#2: When pushing on a part keep your feet well apart (back to front) with the front foot leaning against the base of the machine. This stable position will help to avoid hitting your head on the machine should the blade break.

#3: Never push on a part with your hand in front of the blade. If your part is too small, use a band saw vice or other accessory that keeps your hand well away from the blade.

#4: Thin section parts that are difficult to support (radiators, heat sinks) can be snagged by the blade and should be cut with great care.

#5: In order to avoid pinching your fingers (very painful purple fingernail syndrome) cylindrical parts should be stabilized with clamps or an indexable drill press vice.

#6: To avoid cutting your fingers because of a sliding part, notch the surface to be cut before undertaking an angular cut.

NOW WOULD BE A GOOD TIME TO WATCH FROM A SAFETY PERSPECTIVE:

V-BLOCK PART 7

LATHE

#1: Never leave a chuck key in a chuck! Never, not even for one second!

#2: Never pull on or pick up chips with your fingers, even if the machine is turned off!

#3: A part that is held in a chuck but that is not supported by a tail stock centre must never protrude from the end of the jaws by more than two times its diameter. Short parts however must be held at least one half length in the chuck.

#4: Disengage the feed transmission when using manual feed for cutting. This reduces the wear of the gears and eliminates all possibilities of engaging the feed by mistake.

#5: Long thin parts that protrude from the back end of the spindle must be contained in fixed metal tubes in order to avoid bending the rotating part by centrifugal force creating what resembles a lawn trimmer on steroids that can kill or maim.

#6: Check regularly that your chuck is still well fixed to the spindle.

#7: When polishing with abrasive strips, never let the strip contact the part on more than half its outside diameter. If the strip contacts the full diameter it will snag on the part and quite often **tear your thumbnail off**. When polishing inside diameters, never insert your finger in the hole to be polished. **Never!** Mount your abrasive on a soft wood stick that is notably smaller than the hole to be polished.

#8: Verify the automatic feed's speed and direction before getting close to the part.

#9: For novice machinist, it is preferable to move only one axis at a time, using both hands to move each hand wheel.

NOW WOULD BE A GOOD TIME TO WATCH FROM A SAFETY PERSPECTIVE:

013 HAMMER HANDLE PART 1, LATHE 101

MILLING MACHINE

#1: Milling cutters and fingers aren't made to live together. Handle all cutters with respect, be very careful during installation and removal and never ever let your fingers get close to a rotating cutter. NEVER!

#2: Always verify that the mills rotational direction is set properly for the cutter you are using **before cutting,** because **after** it's too late!

#3: Verify that the tool is properly fixed in the spindle and that all unused axis are locked.

#4: It is dangerous for novice machinists to climb mill. Use conventional or "push" milling until you feel comfortable about your abilities.

#5: Ensure that your part is properly fixed and if you are using a vice, verify that it is well fixed and properly aligned.

#6: Avoid accelerating your automatic feed for tool returns. Many accidents are the result of an operator forgetting to lower the feed before taking the next cut.

#7: Get used to starting the cutter rotation long before the tool gets close to the part to be cut. Large diameter tools or tools with few cutting teeth (fly cutters) can sometimes give the impression of clearing the part when in fact they are in interference.

#8: Never place a waste can, a stool, a chair, a cart or any such object near or next to a milling machine that has automatic vertical feed because it creates a crushing hazard. The same applies to your legs if you are sitting close to the machine (always work standing up).

#9: Check, before starting the machine, that you haven't forgotten to remove the wrench from the end of the spindle draw bar.

#10: Never place your hand under a milling cutter when removing it from the spindle.

NOW WOULD BE A GOOD TIME TO WATCH FROM A SAFETY PERSPECTIVE:

023 THE 1-2-3 BLOCK PROJECT PART 1

SURFACE GRINDER

#1: Dress the grinding wheel regularly.

#2: Verify before starting the spindle that the magnetic table (chuck) is activated.

#3: A part that is higher than it is wide in any direction must be supported when mounted on a surface grinder.

#4: Thin parts do not hold well on magnetic tables.

#5: Most stainless steels do not hold well on magnetic tables.

#6: There should always be a blotter between the grinding wheel and the mounting flanges of your grinder.

#7: The indicated maximum R.P.M. of your grinding wheel must be higher or even to the actual R.P.M. of your machine.

NOW WOULD BE A GOOD TIME TO WATCH FROM A SAFETY PERSPECTIVE:

025 THE 1-2-3 BLOCK PROJECT PART 3

SECTION 3: BASIC METROLOGIE

ACCOMPANYING VIDEOS FOR THIS SECTION:

METROLOGY PART 1

also

METROLOGY PART 2

also

METROLOGY PART 3

3-A: HISTORIC OVERVIEW

Most of our daily activities are controlled by some type of measuring system. The speed limits on our roads and highways are good examples of the impact that measuring systems have on our lives.

One of the oldest recorded unit of measurement is the cubit, which is a linear measurement equal to the distance that separates the elbow from the end of the middle finger. The cubit, as was the case for most units of measurement before the sixteenth century, was based on human anatomy. It became obvious that due to human diversity, it was impossible to have standard and universal measurements

One way of reducing the error due to humane diversity was to average things out. In England, in 1542, the following instructions were proclaimed in order to normalize linear measurements:

"On a certain Sunday as they come out of church, 16 men shall stand in line with the left feet touching, one behind the other. This distance shall be the legal rod and one sixteenth of it shall be the foot."

Averaging out helped, but more precision was needed and averaging had it's limits. Eventually stable objects became the basis for most measuring systems. That is how a stone became the standard for weight measurement and a bronze bar became the standard for linear distance.

Stable objects as standard helped uniformity, but these new standards had their limits. The biggest problem was their uniqueness. Copies where made but even if much attention is taken, a copy is never as accurate as the original and if the original is lost or destroyed, precision suffers.

Today's standards are linked to physical reactions rather than physical objects. Stable and predictable physical reactions solve the problem of unique standards. As an example lets look at the one meter standard that is kept in Paris. The meter we use today can be defined by comparison to that bronze bar (archaic system) or it can be defined by comparing to the length of 1 650 763, 73 wave lengths of red-orange light. The importance of developing accurate standards can be seen in the evolution of industrial production.

- Before the 19 Th century, most parts were unique. This does not mean that two parts could not be similar, it means that they were not interchangeable... if something broke, replacement parts required a lot of adjustment, that is if the parts didn't need to be made from scratch.

- It's during the 19 Th century that specific industries started using "in house measurement standards" making parts produced by that industry or company interchangeable, increasing part quality at the same time.

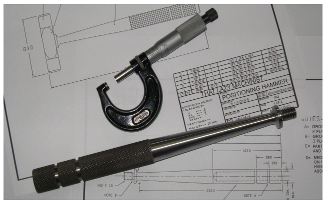
- It wasn't until the early part of the 20 Th century, and the advent of war, that widely used standards became popular. Finally interchangeable parts from different sources, something produced in the United States could fit on something produced in England!

- Today's standards permit a level of precision and interchangeability unimaginable 50 years ago. Our global economy would not exist if not for our capacity to measure accurately based on the same standards worldwide. In actuality, a civilizations technological development is intimately related to it's capacity to measure precisely.

3-B: BASIC TERMINOLOGY AND PRINCIPLES

A: DIRECT MEASUREMENT

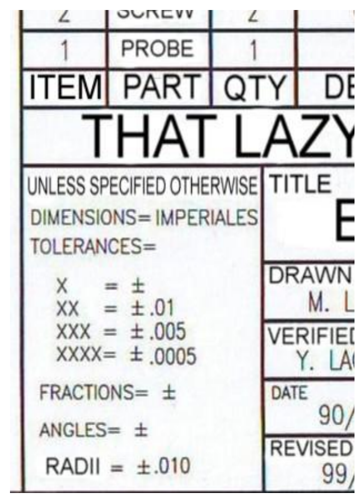
A direct measurement is one that was performed with a measuring tool that incorporates a readable scale. The precision of a direct measurement is directly related to the precision of the instrument used. Steel rulers, micrometers, vernier callipers and protractors are examples of direct measuring instruments.



B: COMPARATIVE MEASUREMENT

A comparative measurement is one that uses an instrument to compare a part to an established standard. This type of measurement is generally considered to be more accurate than direct measurements since it relates to a parts tolerance rather than to a dimension and since no readable scales are used there is less space for interpretation. Sine bars, joe blocks, straight edges and optical comparators are examples of comparative measuring tools.

C: TOLERANCE



Tolerance is the precision to which a dimension indicated on a plan must be respected.

EXAMPLE: 5" +- .005"
The part is acceptable if it's actual dimension is between 4.995" and 5.005" (inclusively)

It is important to note that measuring tools also have tolerances.

EXAMPLE: A standard inch micrometer should not be used to measure a part having a tolerance of +- .0005" or less

D: STANDARD

Object or reaction that is the material representation defining a measuring system.

EXAMPLE: The standard for time is the vibration of a caesium atom, 9 192 631 770 vibrations of this atom is equal to one second.

E: SURFACE FINISH

Description and classification of surface imperfections produced by machining operations.

F: DECIMAL PRECISION

On a drawing or a sketch, it is important to use the correct number of decimals on a dimension since they generally indicate to the machinist the precision required for measuring.

EXAMPLE: Imperial steel ruler = two decimals
Imperial vernier calliper = three decimals

G: ERROR MULTIPLICATION

It is crucial to avoid moving you reference point when measuring. It is much more accurate to measure one four foot length than four one foot lengths. Each measurement includes an error. Stringing together several measurements compromises accuracy.

H: IMPERIAL SYSTEM (INCH)

In the machine shop, the unit used for linear imperial measurement is the inch. In the shop we prefer saying Twenty five inches rather than two feet and one inch.

The imperial system can however be fractional or decimal.

The fractional systems stems from a time when tradesmen where poorly trained in mathematics and calculators did not exist. The reason being that it is easier to perform basic math operations on fractions. Today, the fractional system is mostly used to define nominal stock dimensions or accurate tool dimensions.

The decimal imperial system is based on the thousandth decimal.(0.001"). In the shop we describe decimal imperial dimensions in inches first and thousandths of an inch second. So 3.105" would be described as three inches, one hundred and five thousandths. If ten thousandths of an inch are present they are stipulated separately at the end.

EXAMPLE: 2.0533" would be described as two inches, fifty three thousandths and three ten thousandths.

I: METRIC SYSTEM (MM)

In the machine shop, metric dimension are described in millimetres. It is preferable to say 25 millimetres rather than two point five centimetres.

when a dimension needs to be described more accurately, we still describe it in millimetres.

EXAMPLE: 30,51 mm would be described as thirty point 51 millimetres.

Please note that in Europe the decimal point does not exist in the metric system, it is replaced by the comma.

3-C: LINEAR MEASUREMENTS IN THE SHOP

More often than not, measurements in the shop are linear measurements. Parallel surfaces, inclined surfaces and profiled surfaces are all measures with linear measurement techniques from a reference surface or reference point.

Example: In order to measure a cylindrical shape, we measure its diameter. Even if the surface is curved, it is measured in a linear fashion. If the surface is an irregular three dimensional curve, it will still be measured linearly with X,Y and Z Cartesian coordinates.

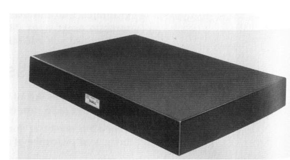
There are a multitude of different parts and shapes to measure in the shop and that is why there are so many different types of measuring tools. Some measuring tools are very accurate others are much less, what is important is to choose the proper measuring tool for the job at hand.

The most important thing to consider when choosing a measuring tool is the precision (tolerance) of the part being measured. It might be tempting to choose the most accurate measuring tool for all jobs. However a person using a vernier micrometer (accurate measuring tool) to measure a rough casting is possibly damaging an expensive tool when an inexpensive steel ruler would do the trick just as well since the rough casting has no accurate surfaces to measure from.

Another important factor to consider is availability. Since measuring tools are expensive, it is rare that a shop has all possible tools in inventory. A good machinist finds ways to get good result by imagining different ways of measuring hard to reach entities. Regardless of how accurate a tool is, the end measurement will only be good if the machinist uses it properly. Measuring parts isn't difficult! What is difficult is getting good results that reflect the tolerance of the part being produced.

The basic linear measurement tools are:

SURFACE PLATES
STEEL RULERS
VERNIER CALIPERS
MICROMETERS (int., ext., depth)
VERNIER HEIGHT GAUGES
DIAL INDICATORS
GAUGE BLOCKS



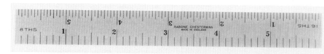
<u>**SURFACE PLATES**</u>:

Surface plates are very flat and rigid tables made of granite or cats iron that are a reference surface for accurate measurement, assembly and layout. Surface plates are massive but they are to be used delicately since they are one of the most accurate measuring tool in the shop.

SURFACE PLATES ARE NOT WORKBENCHES!!!

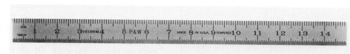
STEEL RULERS:

Steel rulers are one of the most used measuring tools in the shop. They are available in many sizes and are quite versatile when used with ruler accessories. A precision steel ruler can be read to with an accuracy of 0.010". Imperial precision rulers are generally graduated in fractions (1/32"and 1/64"), in decimal (0.010") and metric rulers are usually graduated in half millimetres.



Flexible steel rule (fractions of an inch).

Flexible steel rule (decimal inch).

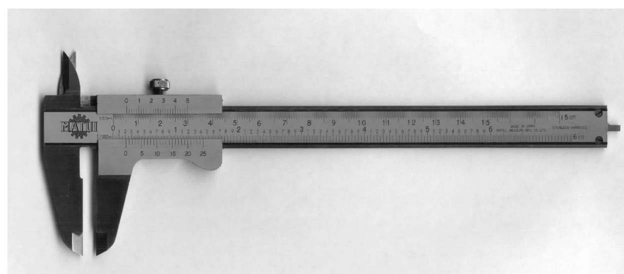


Flexible steel rule (metric).

Standard hook rule.

VERNIER, DIAL AND DIGITAL CALIPERS:

Vernier callipers are precision measuring instruments used to measure parts within a tolerance of .002" or 0,04 mm. They can be had with a vernier scale, a dial scale or a digital readout. The tools accuracy is affected by its quality but not by the type of scale used. Most measuring callipers come equipped for external, internal and depth measurements.



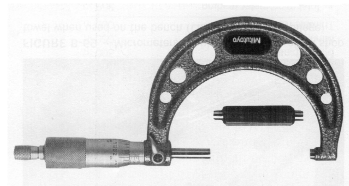
VERNIER CALIPERS 101

MICROMETERS:

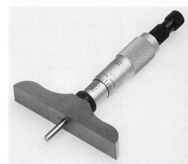
Micrometers are the most accurate of the everyday direct measuring tools. A standard micrometer can measure with a precision of 0,02 mm or .001". A vernier or digital micrometer can measure with a precision of 0,002 mm or .0001". Obviously this supposes that you are using good quality tools and that you are using them properly.

Micrometers are very accurate but they are also very limited. A micrometer can measure only one type of surface and in 1" or 25 mm increments. You must use a 0 to 1" micrometer in order to measure a 0 to 1" part and so on. That means that you would need to have several (at least 8) micrometers in order to perform all the measurements that a 6" vernier calliper can.
There are three major families of micrometers:

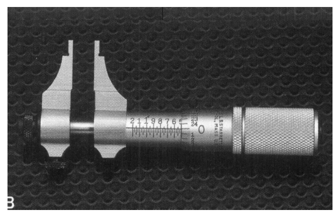
EXTERNAL MICROMETERS



DEPTH MICROMETERS

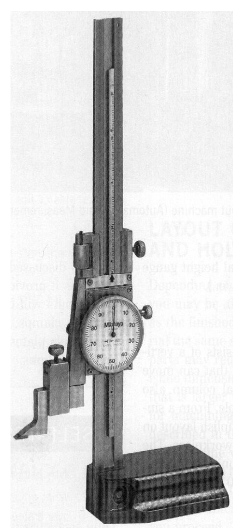


INTERNAL MICROMETERS



MICROMETERS 101

VERNIER, DIAL AND DIGITAL HEIGHT GAUGES:



Height gauges are very versatile tools. They can be used for direct or comparative measurements and as a layout tool. Height gauges resemble vernier callipers mounted vertically on a very accurate base. The base stabilizes the tool when used in conjunction with a surface plate and permits very accurate measuring.

DIRECT MEASUREMENTS: With a good vernier height gauge it is possible to perform linear measurements within .001" or 0,02 mm and digital height gauges can be even more accurate.

COMPARATIVE MEASURMENTS: When used in conjunction with a dial indicator and gauge blocks, the height gauge can be used for extremely accurate comparative measurements.

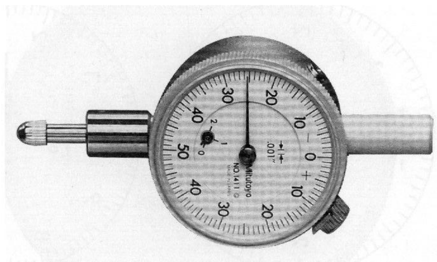
LAYOUT: A point for scribing can also be mounted on the height gauge for accurate layout.

DIAL INDICATORS:

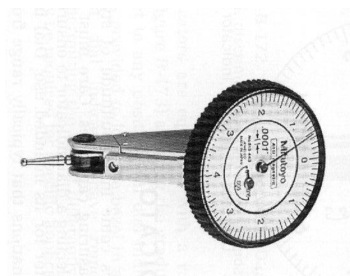
Dial indicators are mostly used in conjunction with gauge blocks as comparative measurement tools. They are also used for tool and machine alignment such as milling vice truing and part centring in a lathe four jaw chuck. In some cases they can be used as direct measuring tools.

There are two major types of dial indicators:

CONTINUOUS-READING DIAL INDICATORS:



PRECISION TEST DIAL INDICATORS:



ACCOMPANYING VIDEOS FOR DIAL INDICATORS:

TRUING A MILLING MACHINE VICE, MILLING 101

ALSO

FOUR JAW CHUCK CENTERING, LATHE 101

ALSO

HEAD ALIGNEMENT AND EDGE FINDING, MILLING 101

GAUGE BLOCS:

Gauge blocks are very hard and stable blocks that are very very accurate. The two measuring surfaces of each block lapped and polished to an accuracy of +- 0,00005 mm for the most accurate sets. The blocks accuracy is affected by temperature so they must be used at 20º C (68°F)

There are three classes of gauge blocks:

AA: Generally used to verified the two other classes of blocks.

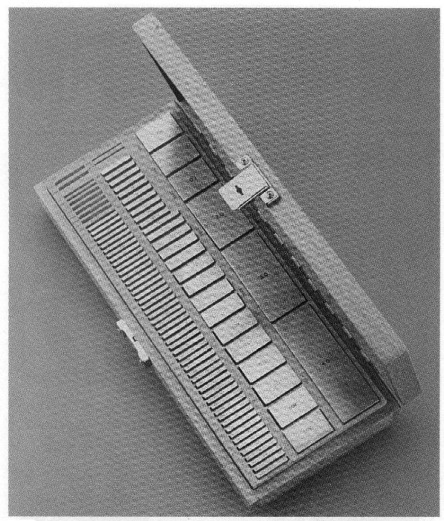
+- 0,00005 mm

A: Generally used to calibrate measuring tools.

+ 0,00015 mm - 0,00005 mm

B: Generally used for accurate measurements in the shop.

+ 0,00025 mm - 0,00015 mm



GAUGE BLOCKS & SINE BARS 101

3-D: HOW TO READ A VERINIER SCALE

Vernier scales use ***asymmetrical scales*** to amplify what would normally be too small to see.

The imperial vernier calliper has two graduated scales. The first is the fixed scale and is graduated in 0.025" segments. Four segments of the fixed scale equal 0.1" and 10 divisions of 0.1" equal 1".

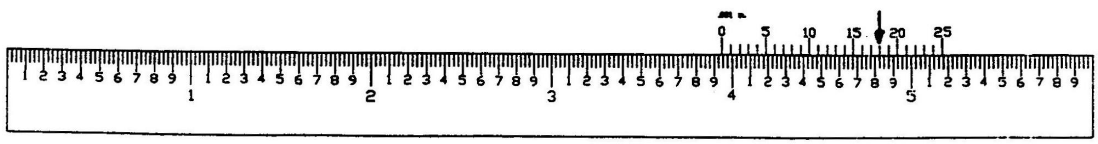
The second scale is the vernier scale and it is divided in 25 segments. Each segment **REPRESENTS** 0.001". These divisions don't measure 0.001"!!! This 25 divisions are equivalent (in value only) to one division on the fixed scale.

In order to read an imperial vernier calliper you must first count all the **complete** divisions visible before the **zero line** of the vernier scale. This first reading will always be a multiple of 0.025". The reading for the fixed scale of example **A** is 3.925". The reading for the fixed scale of

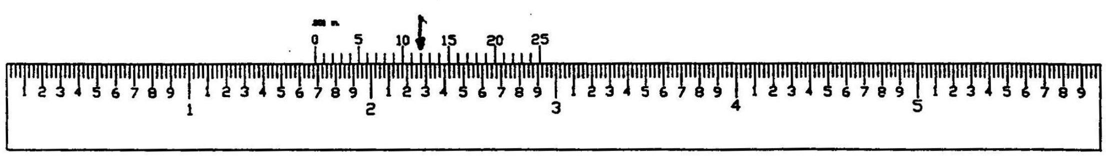
example **B** is 1.675".
Once you have noted the reading for the fixed scale you must find the line on the vernier scale that aligns with a line on the fixed scale. Unless the dimension of the part falls on a multiple of 0.025" (in which case the vernier scale will not be needed) the vernier scale will read somewhere between 0.001" et 0.024" inclusively. The reading for the vernier scale of example **A** is 0.018" and the reading for example **B** is 0.012".

The final measurement is obtained by adding the result of the fixed scale to that of vernier scale

Example **A**: 3.925" + 0.018" = 3.943"



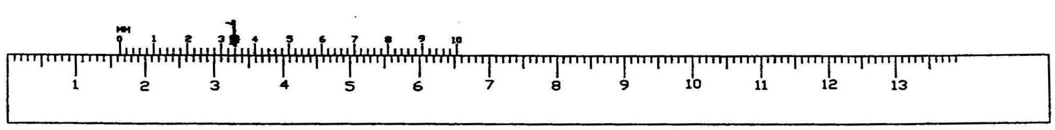
Example **B** : 1.675" + 0.012" = 1.687"



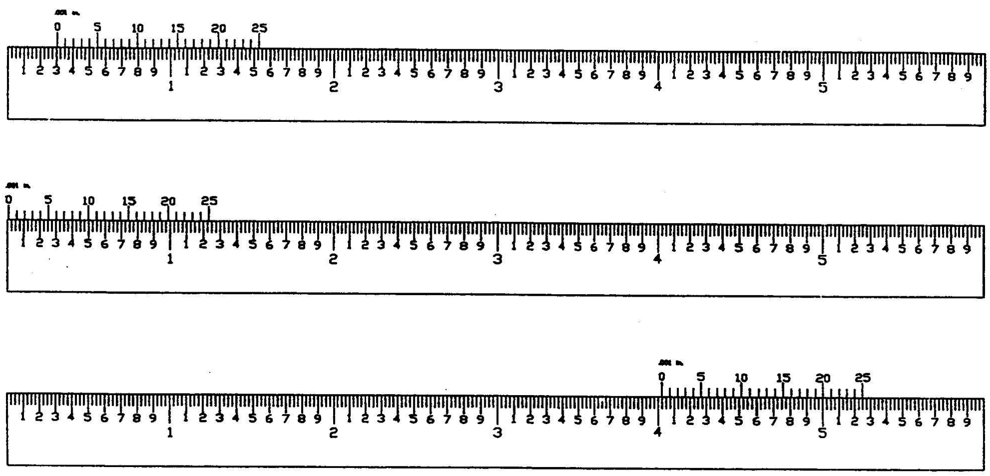
A metric vernier calliper also has two scales the longest is the fixed scale and it is graduated in 1 mm segments every tenth segment representing 1 cm.

The second scale is the vernier scale and is graduated in 50 segments **REPRESENTING** 0,02mm each**.** The 50 divisions of the vernier scale are equivalent to one division on the fixed scale.
In order to read a metric vernier calliper you must first count all of the **complete** divisions on the fixed scale up to the **zero line** of the vernier scale. This first reading will always be a multiple of 1 mm. The fixed scale of example **C** shows 16 mm and the fixed scale of example **D** shows 53 mm. You must then add to the result of the first reading the amount shown by the line of the vernier scale that aligns with a line on the fixed scale. This second reading will always be a multiple of 0,02 mm. The final dimension obtained by adding the first reading to the second. The reading for the vernier scale of example C is 0,34 mm.

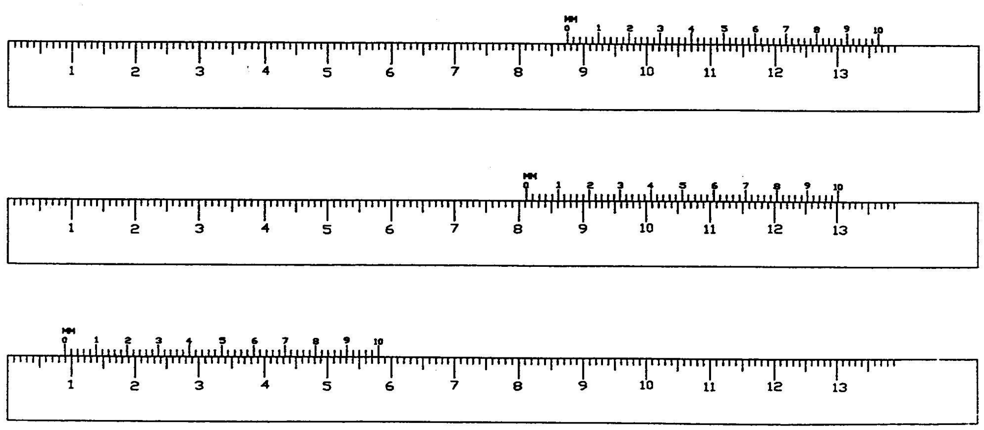
Example **C** : 16 mm + 0,34 mm = 16,34 mm



A few examples of imperial vernier scales:



A few examples of metric vernier scales:



Associate dimensions with scales: **0.006" 81,18mm 87,46mm 0.304" 8,94mm 4.023"**
remember that the lines on an actual vernier scale are much crisper than these scans.

3-E: ANGULAR MEASUREMENT IN THE SHOP

With the exception of profiled surfaces, angular surfaces are the most difficult to measure accurately since angular measuring tools are generally more difficult to manipulate and since angular measurements depend on the interaction of many variables.

> EXAMPLE: A standard micrometer is the only tool required in order to measure a cylindrical part within +- 0.001". With only a micrometer I can easily verify the parts diameter, cylindricity and its parallelism. In comparison, it is not possible to verify a surfaces flatness with a protractor. In order to verify the flatness of an angles surfaces, you need other tools and as we know multiple measurements for a single entity (error multiplication) increases the possibility of error.

The most common angular measurement tools are:

> PROTRACTORS.
>
> VERNIER PROTRACTORS.
>
> SINE BARS AND GAUGE BLOCS.
>
> OPTICAL COMPARATORS.

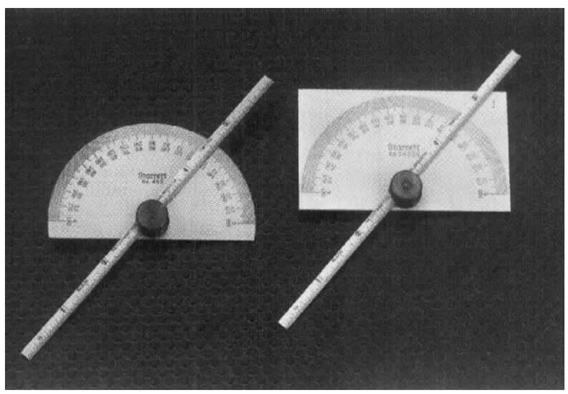
TRIG & TRIANGLES SHOP MATH 101

also

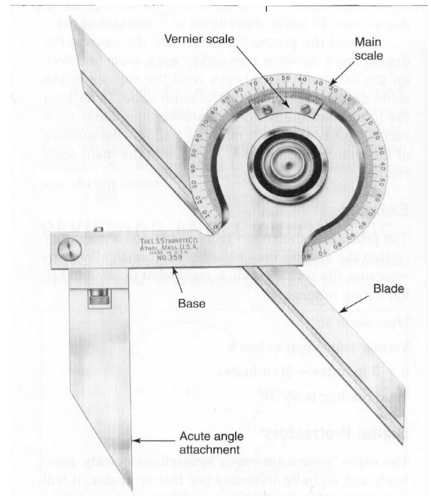
EMMA'S ANGULAR PROBLEM!

PROTRACTORS:

Simple protractors are used for low precision angular measurements. Their scales are generally composed of 180 divisions of 1º each. Even if these tools are not very accurate, they are often used to approximate an angle before using a sine bar.

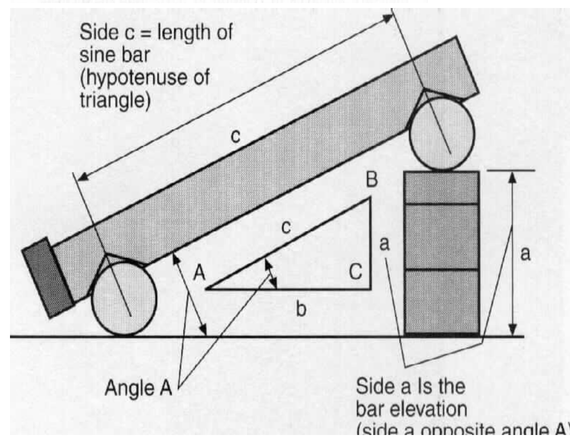


VERNIER PROTRACTORS:



Vernier protractors are used for angular measurements of medium accuracy. These tools can be read to an accuracy of +- 5' (+- 5 minutes of a degree) which is sufficient for most angular measurements in the machine shop. There are two (at least) measuring surfaces on these tools. The first is the foot which incorporates the fixed scale that consists of 360 division equal to 1° each. The second is the ruler that is attached to a vernier scale of 24 divisions each **representing** 5' of a degree. These protractors come equipped with several attachments that permit a wide range of angular measurements.

SINE BARS:



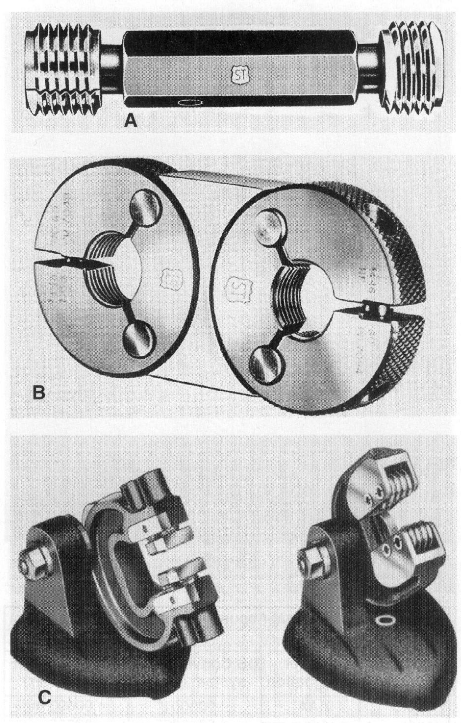
Sine bars are the most accurate general purpose angular measuring tools used in the shop. However, it must be used in conjunction with a surface plate, gauge blocks and a dial indicator that is usually mounted on a surface gauge. This makes for a very expensive method for measuring angles and should only be used when required. Trigonometry is used to set its height and the variables needed are either height of the opposite side of the angle to be measured (required gauge blocks) or the angle itself. The hypotenuse of the right angle triangle formed will be the distance separating the two cylindrical support surfaces. This distance and the diameter of the cylinders are extremely accurate.

3-F: GAUGES:

Many people are use to looking at measurements as being absolutes. Machinists know that that is not the case and that all measurements have tolerances and limits within which the dimension is deemed correct. This does not imply that parts with a loose tolerance (example +-0.01") are inaccurate. All parts that respect the tolerance indicated on the blue print are equally accurate whether the tolerance is tight (example +-0.001") or loose. Tolerances must reflect the end use of a part, making a butter knife as accurately as a surgical scalpel would be a great waste of time and money.

Gauges exploit the fact that all dimensions have tolerances to accelerate inspection and lower the cost of quality control.

Gauges are comparative measuring tools and most gauges give no indication of the actual dimension of a part. All they do is indicate whether a part is within the acceptable tolerance.



Since all dimensions have an upper and lower limit it is normal that two gauges be required to indicate if the part is acceptable. These upper and lower limits gauges are known as **"GO AND NO GO GAUGES"**. If one end of the go and no go gauge shown in example **A** threads into a hole but the other end does not, the thread is within its tolerance. If neither end thread in or if both ends thread in, the part is out of tolerance.

Gauges are very accurate tools but they also have a tolerance. Usually a gauges tolerance must be around one tenth the tolerance of the part being measured.

Example: **Tolerance of the part +- .001"**

Tolerance of the gauge +- .0001"

Many gauges are commercially available but an experiences toolmaker can produce special gauges for specific jobs.

SECTION 4: BENCHWORK

4-A: INTRODUCTION

In the shop, the difference between an OK part and an EXCELLENT part is quite often related to proper bench work techniques.

For the novice machinist it is important to become adept at bench work as soon as possible since it is a good way to increase ones awareness of the properties of the material being cut. When using bench work tools, the machinist is almost in direct contact with the chip production (cutting) and that helps to understand how metal cutting works.

All you have to do is open a tool supply catalogue to see that there are many different types of hand tools. Since this lesson is for novice machinists we will only look at the most common bench work tools.

ACCOMPANYING VIDEOS FOR BENCH WORK:

BENCH WORK PART 1

ALSO

BENCH WORK PART 2

ALSO

BENCH WORK PART 3

ALSO

BENCH WORK PART 4

ALSO

BENCH WORK PART 5

ALSO

BENCH WORK PART 6

ALSO

KEYWAYS & BROACHING

ALSO

DRILL POINT GAUGE PART 1

ALSO

DRILL POINT GAUGE PART 2

ALSO

DRILL POINT GAUGE PART 3

ALSO

DRILL POINT GAUGE PART 4

4-B: VICES:

Vices are used to immobilize and position parts so that they may be cut or machined accurately. Since many bench work operations require that both hands be on the tool, we can say that bench vices often serve as a third hand. Since there are many types of parts and many types of operations to perform, it is not surprising that there exist many types and sizes of vices. The three main types of vices are: **Bench vices**, **drill press vices** and **milling machine vices**.

BENCH VICES:

Bench vices are available in many different sizes and weights. They are to be properly secured to a work bench and are used to stabilize parts for assembly, punching, hack sawing, filing, polishing, threading ... They are usually mounted so that their fixed jaw is just proud of the edge of the work bench as to permit the holding of long parts vertically.

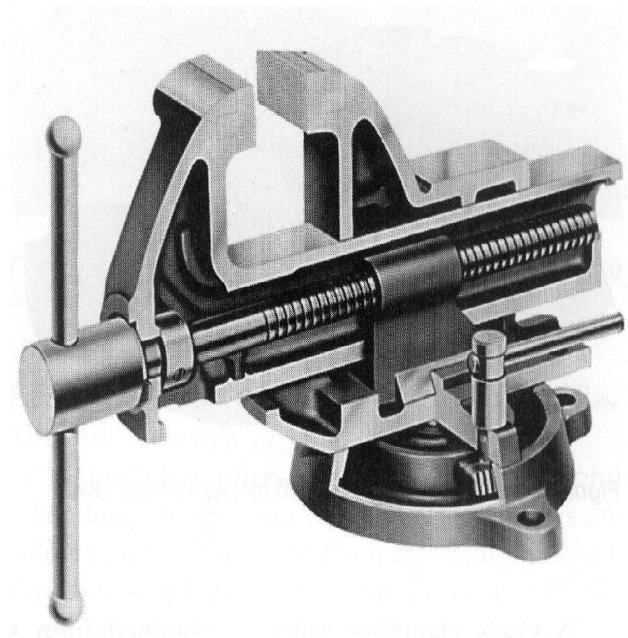
Characteristics of a good bench vice:

- A strong and rigid structure, a good vice is made of forged medium carbon steel (best) or ductile cast iron (acceptable).

- The vice jaws should be removable in order to replace when damaged or to fit special jaws for specific parts (soft jaws, v-grooved jaws...).

- A 360º rotary base.

- An Acme drive thread and if possible a split nut quick release in order to avoid wasting time spinning the handle.

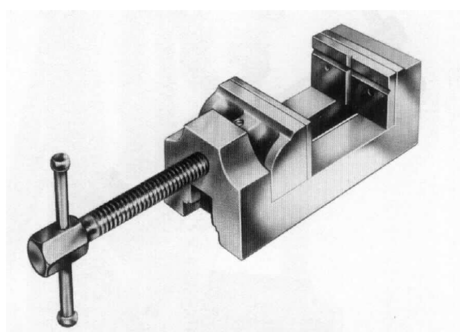


N.B. A bench vice is not an anvil !!!

DRILL PRESS VICES:

These vices have wide and stable bases which is important since they are often deposited on drill press tables with out being fixed to them.

They are usually made of grey cast iron since the lateral forces exerted on them are quite low and since this material has excellent anti friction qualities when sliding on steel. Grey cast iron is quite inexpensive which makes these vices quite affordable and that is good since they are the most abused and ill loved vices in the shop. For ease of manipulation these vices are designed to be very light



and that also makes them quite delicate and not very resistant to shock.

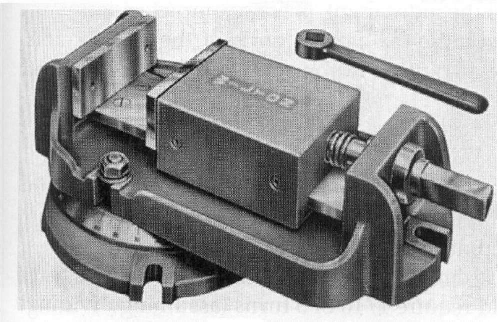
N.B. Some drill press vices can be used on their sides and even upside down. These indexable drill press vices (photo) can also be used (upside down) as vertical band saw vices as seen in the

MACHINE SHOP SAFETY PART 3

MILLING MACHINE VICES:

These vices are usually bolted firmly to that machine tools work surface. They are very rigid and can exert a huge amount of holding force on the work part since the machine tools that they are used with can exert a lot of lateral force on the part during the machining process.

They are massive rigid and very accurate since they must be at least as accurate as the machine tool that you are using them on. Do not be fooled by their size, these vices are and must remain very accurate and must be handled with great care.



Some machine vices incorporate a rotary base (for angular cuts) and or a pivot axis (for compound cuts). Rotary base vices and universal vices should only be used when required since the rotary axes reduce rigidity and make for a weaker setup.

N.B. Although one could use a dead blow hammer to loosen a machine vice's handle, it should never be used to tighten one. The vice handle has been designed for the vice and handle extensions should not be used so if the vice is having trouble holding the part get a bigger vice!

4-C: HAMMERS

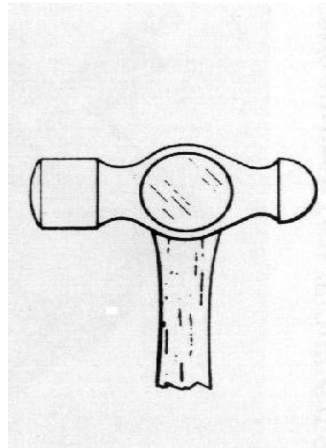
Most people know that hammers are made to strike (hammer) things. The problem is that that is all that most people know about hammers.

It is important to know that hammers are complex tools that are designed for specific jobs so it is normal to think that there are many many types of hammers. Here are the three basic types used in a machine shop.

BALL PEEN HAMMER:

These hammers are available in many different weights and each hammer has two very different faces (striking surfaces). The first face is slightly convex and is used for punching, assembly and forming operations. The second face (the peen) resembles a half sphere and is generally used for riveting and peening operations.

Here are a few suggestion for proper and safe use of a ball peen hammer:



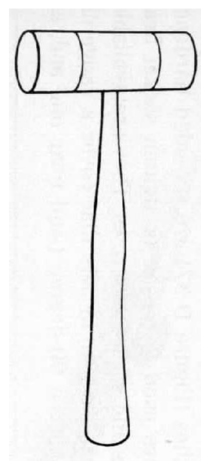
- Other than when peening, the hammer's movement must not be pendulum like. This hammer should be dropped (straight vertical movement) rather than swung.

- It should not be used on materials harder than 50Rc.

- Hammers with loose heads or damaged handles should never be used.

- If you need to hit harder, choose a heavier hammer! Always use a safe striking speed. Use gravity to accelerate the hammer, not muscle.

POSITIONING HAMMER:

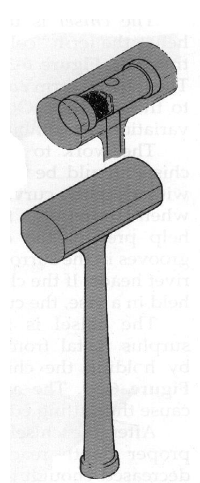


Lite and delicate, these hammers are designed for crisp energy transfer when positioning without marring the surface being struck.

They are used when positioning work parts on machine tools. Rapid energy transfer versus a parts capacity to absorb it is the corner stone of accurate positioning.

N.B. The hammer produced in the hammer project videos is a positioning hammer.

DEAD BLOW HAMMERS:

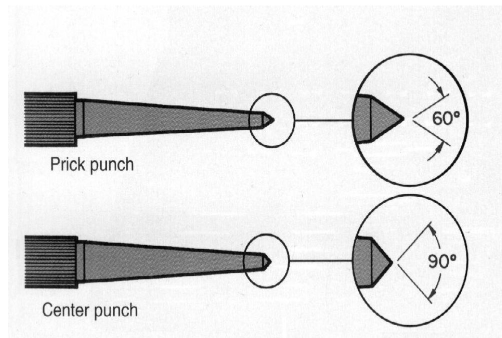


These hammers are massive and are used to seat parts in machine vices thanks to an internal cavity that is half-filled with small steel balls. The steel balls movement when striking the part reduces the rebound problem by stretching the strike time thus absorbing the excess energy that would normally make the part bounce of the bottom of the vice. I guess we could say that when action meets immobility, we have rebound and when seating a part rebound is not good.

4-D: PUNCHES

PRICK PUNCHES:

These punches have 40º to 60º points and are used to produce small indentations on the surface of a part in order to position one of the points of a divider or to make an accurate starting point for a centre punch or for producing a permanent layout.



N.B. Since punch penetration is a factor of surface of contact between the punch and the part, it is recommended to use a lite ball peen hammer with these punches.

CENTER PUNCHES:

These punches have a 90º points and are used to produce wide indentations that are used to guide drills to a specific position during a hand positioned drilling operation.

For drills that are 1/4" or smaller, the drill can be started directly in the indentations. A centre or spot drill will be used before using drills larger than 1/4".

LETTER AND NUMBER PUNCHES:

These punches are mostly used to identify parts. It is crucial to practice with these punches since poor lettering will deter a client's attention and make dimensionally perfect parts unacceptable. Always remember that precision can be measured but perception is just as important since many clients depend on a part's appearance to determine its acceptability. Weird but true!

N.B. It is important to never punch parts that have been hardened, plated or anodized.

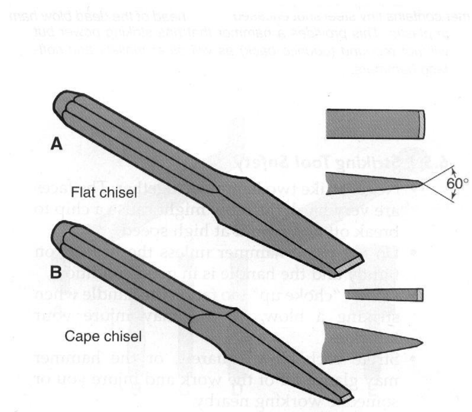
DRIFT PUNCHES:

These cylindrical punches have flat ends and are used for extracting dowel pins, split pins and stops that are assembled using light press fits.
It is important to use many small crisp blows from a lite weight ball peen hammer (same principal as

an impact wrench) rather than massive blows from heavier ones. When pins cannot be extracted with a reasonable force, penetrating oils, presses, heat or pullers should be preferred.

COLD CHISELS:

A cold chisel is a punch that is designed for cutting metal. In pre milling machine days, these tools were used to cut sheet metals as well as surfacing and grooving parts. Today, chisels are mostly used to behead screws or remove rusted nuts that cannot be removed by conventional means. They are also used for swaging tongues for tongue and groove assembly.



N.B. The end that comes in contact with the hammer will gradually mushroom. It is imperative to remove any mushrooming with a grinder. It is recommended to use a heavy leather glove or a plastic hand guard when using a chisel since your hands are not hammer proof.

4-E: FILES

Files are unidirectional chip producing cutting tools! They are not abrasives and cannot cut on the back stroke.
There exist a multitude of different shapes of files (flat, half round, knife edge, round, rat tale, rifler .....) since the file should conform to the shape of the surface that will be produced.

A file's surface is made up of a large number of teeth. These teeth incorporate a cutting surface and cutting lips that will engage with the part and produce chips by plastic deformation. Files are unidirectional cutting tools, they cut from the tip of the file to the handle and no pressure should be applied on the backstroke.

N.B. File are made from a very brittle tool steel and will shatter if abused. This is very dangerous for your eyes.

Never use a file as a pry bar!

Never use a file that does not have a proper handle

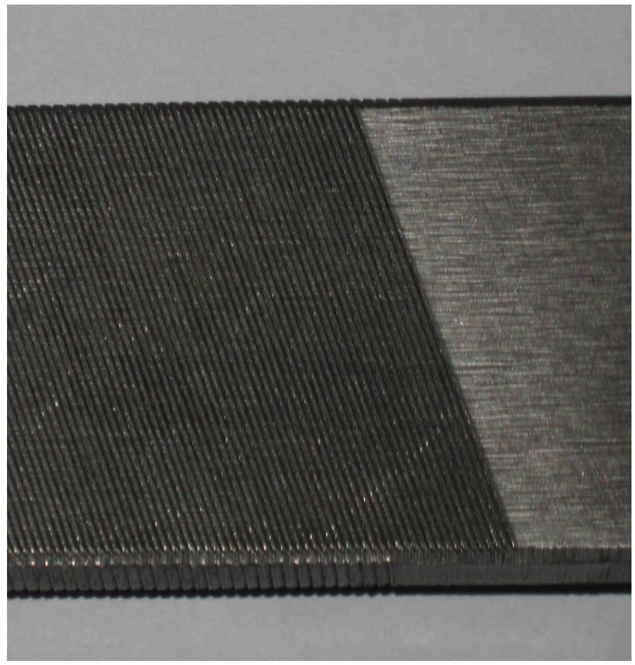
Do not hit the file against a hard surface to clean it! Always use a file card.

FILE TOOTH FORMS:

The choice of a file shape is quite simple. It must conform to the surface of the part being produced. However the choice of the file's tooth form is much more complex.

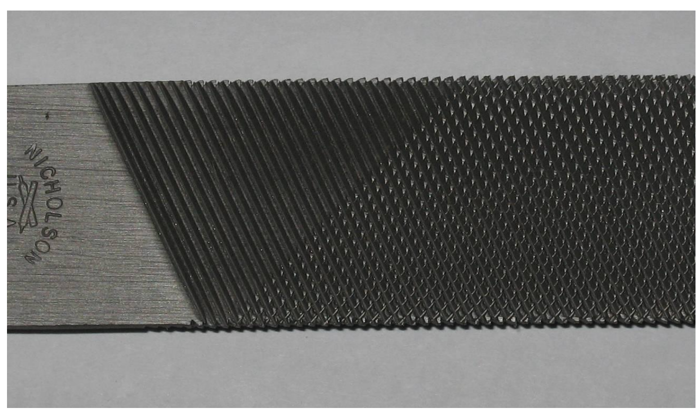
Since this is an introductory course, we will only look at single cut and double cut files.

SINGLE CUT FILES have many parallel rows of teeth that are inclined by about thirty degrees from the sides of the file. This tooth form is generally used for finishing operations since their lateral and progressive cutting actions create long chips that tend to produce a better surface finish. This tooth form is also preferred when filing hard materials (not more than 50Rc). This tooth form is however not suited for rapid metal removal.



N.B. Some single cut files have teeth that are inclined at sixty degrees from the sides of the files. These files are specifically used for filing on the lathe. These files are difficult to control for bench work because of the lateral thrust that comes with such a steep tooth angle. However when filing on the lathe, the steep tooth angle clears the chips better and reduce material incrustation.

DOUBLE CUT FILES have diamond shaped teeth that are the result of two intersecting rows of teeth similar to the single cut ones. This tooth form is generally used for roughing cut since the smaller contact area permits better penetration so they can take deeper cuts. The space between the teeth also permits for more chip clearance which is helpful when filing large surfaces.



TOOTH PITCH:

The size of a file's teeth is a very important when choosing a file. Three factors must be considered when selecting a tooth size.

#1: **The type of work can vary.** A job that entails the removal of a lot of material requires the use of large teeth (roughing). A job that entails the removal of a small amount of material calls for the use of small teeth (finishing).

N.B. In order to work efficiently it is normal to use several tooth pitches to complete a job (from roughing to finishing).

#2: **La The hardness of the material** affects the selection of a tooth pitch. It is best to choose small teeth for hard materials and large teeth for soft materials.

#3: **The size of the filed surface** also affects tooth size. Large teeth should be used when filing large surfaces since better chip clearance is required (the chips must be stored in the file from the tooth's engagement with the part until the tooth exits the other side.

N.B. As you can see, the factors that guide tooth pitch selection are often contradictory. There is no pitch size that will remove material efficiently from a large surfaced hard part. Choosing the proper file for a job requires compromise.

FILING TECHNIQUES:

There are three basic filing techniques. The most popular is the "filing is easy" technique. This technique requires no training and rarely gives good results. The two other techniques are draw filing (finishing) and cross filing (roughing).

DRILL POINT GAUGE PART 1

also

OILY FILES?

4-F: HACK SAWS

We all know that saws are made to cut things. It is however important to remember that they do not cut parts in two, they cut them in three, the two parts that result from the removal of the third (the chips). Always take into consideration the kerf (width of cut) when calculating the amount of stock required for a job.

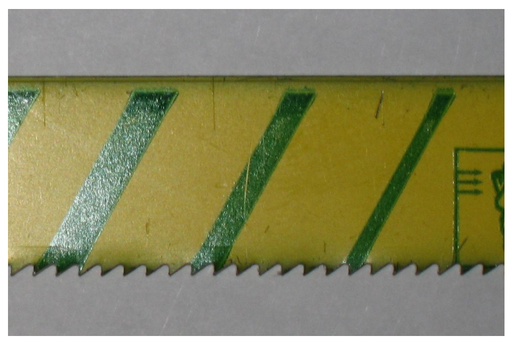
There are three basic types of hacksaw blades.

#1: **High carbon steel blades** are low quality blades generally reserved for sawing mild metals such as brass, copper, aluminium, mild steel ...

#2: **Bi-metal blades** have high speed steel teeth welded to a hardened and drawn medium carbon steel body. These composite blades have hard and wear resistant teeth and flexible and forgiving (will not break easily when flexed) blade bodies. The are mostly used to cut tougher and harder (45 Rc max.) material such as hardened medium carbon steels, stainless steels, bronzes ... These blade are however two to three times more expensive than the plain high carbon steel ones.

#3: **Precision hacksaw blades** are integrally constituted of high speed steel and are thicker than regular blades and do not have any blade set (the teeth are the same width as the blades body). This structure makes for a very stable blades that track very accurately and that produce dimensionally accurate kerfs (width of slot). However, these blades are very expensive and quite easy to break. They are mostly used for accurate grooving operations and are not recommended for general sawing.

BLADE PITCH:



Blade pitch is defined as the number of teeth per inch of blade. A #18 blade (the go to general purpose pitch) has 18 teeth per inch. The higher the number, the smaller the teeth.

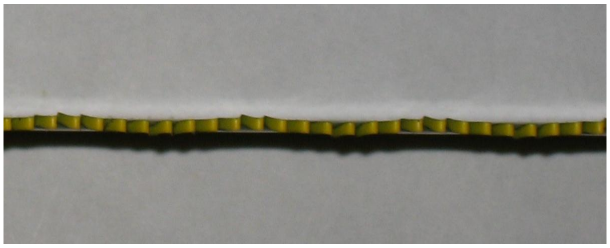
Two factors are to be considered when selecting the proper blade pitch for a job.

#1: **The thickness of the part** to be cut affects the choice of blade pitch since it defines the quantity of material that will be stored between the teeth from the start to the end of a single cut stroke. **The thicker the part, the bigger the teeth!**

When the part is thin, it is important to choose a blade pitch that ensures that **at least two teeth are in contact with the blade at all times**. This ensures that the part cannot fit between two teeth, since this could sheer teeth off of the blade.

#2: The parts hardness also affects the selection of blade pitch. The tooth's size selection is inversely proportional to the part hardness. **The harder that part, the smaller the teeth**.

BLADE SET:



The blade set is responsible for the difference between the width of the blade's body and the width of the kerf (slot) that it produces. In order to prevent the blade from snagging (binding in the kerf) the teeth are offset from side to side in order to cut a little wider than the blade's body. Blade offset reduces friction but makes the blade difficult to control and keep cutting straight. This is why precision hacksaw blades have no offset. It is the difference between the blade set and body height that defines the smallest possible radius that can be produced with a given blade.

A FEW SUGGESTIONS:

- #18 bi-metal blades are the general purpose blades preferred for most jobs.

- It is important to use the full length of the blade when cutting.

- Hack sawing should not require a lot of effort. Let the blade cut, pushing or forcing it will reduce blade life and will make it very hard to cut straight.

- General use hacksaw blades are designed for straight or slight radius cuts. For pronounced curves use a cope or scroll saw.

- Cutting speed:

    Softer metals: Two blade lengths per second.

    Harder metals: One blade length per second.

- Saw only with your arms, your body should not move.

- Always cut vertically. For angular cuts, reposition the part to permit vertical cutting.

- Never insert a new blade in the kerf produced by a worn blade.

DRILL POINT GAUGE PART2

4-G: TAPS AND DIES

Taps (internal threads) and dies (external threads) are thread cutting tools. Some are designed for bench work and others for use on machine tools. Although not covered here, threads can also be produced by rolling, by single lip lathe cutting and by grinding.

Before describing these tools, it would be good to define what threads are and their basic terminology.

WHAT IS A THREAD?:

A thread is a helical groove of constant cross section on the outer surface of a cylinder (external thread) or on the inner surface of a bore (internal thread). Threads can be used for assembling parts, for accurate movement, for measuring and as a force multiplier (mechanical advantage).

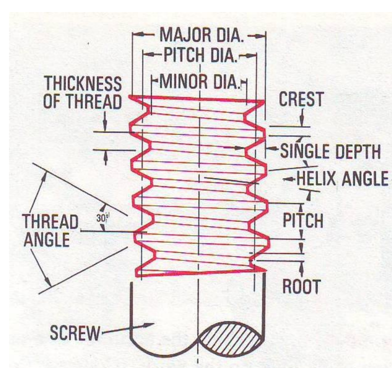
THREAD FORMS:

There are several different thread forms Since this is an introductory course, we will limit ourselves to the three basic (most common) forms: Unified (imperial), ISO (metric) and Acme (Wile E Coyote).

Unified and **ISO** threads have **60 degree V** grooves and are generally used for assembly. They can also be used for accurate measurement (as is the case for micrometers).

ACME threads have **29 degree V** grooves and are generally used for accurate movement or for mechanical advantage and quite often for both (lead screw on milling machine table).

THREAD TERMINOLOGY



- **THE MAJOR DIAMETER** (sometimes referred to as the outside diameter) is the largest diameter of an external or internal thread.

- **THE NOMINAL DIAMETER** **"represents"** the major diameter when expressed as the closest (larger) fractional dimension for imperial threads or closest (larger) millimetre for metric threads. It is not the actual dimension of the outside diameter but rather a simplified way of presenting the thread. It would be awkward to describe a **1/2"-13 2A UNC** thread as a **0.4985" to 0.4822"-13 UNC** thread. It is much simpler to just express the OD as 1/2" since the actual major diameter varies depending on the class of fit required.

- **THE MINOR DIAMETER** (sometimes referred to as the root diameter) is the smallest diameter of an external or internal thread.

- **THE PITCH DIAMETER "represents"** an imaginary cylinder that passes through the midpoint of the threads depth. The pitch diameter is used for calculating the measurement over the wires for accurate thread production.

- **THE PITCH** of a thread is the distance that the thread travels axially in one full turn. Be aware that the pitch is not the number of threads per inch.

- **THE THREAD ROOT** is the bottom of the V groove. For internal threads the root is on the major diameter and for external threads the root is on the minor diameter.

- **THE CREST** is the top of the V groove. Fort internal threads the crest is on the minor diameter and for external threads the crest is on the major diameter.

-**THE DEPTH OF THREAD** is the radial distance that separates the root from the crest of a thread.

- **THE HELIX ANGLE** (often referred to as the lead angle) is the angle of the thread groove in relation to the thread's axis. A coarse pitch thread (large thread) has a high helix angle and a fine pitch thread has a low helix angle.

<u>THREAD NOMENCLATURE</u>:
Unified threads smaller than 1/4" dia. are described in this manner:

#8 - 36 UNF

←	↓	→
Nominal code (# x.013)+.060	Number of threads per inch	Identification of thread form and series

Unified threads of 1/4" dia. or larger are described in this manner:

1/4 - 20 UNC

←	↓	→
Nominal dia. diameter	Number of threads per inch	Identification of thread form and series

ISO threads are described in this manner:

M 10 X 1.5
← ↓ →
Metric Nominal dia pitch

N.B. Many other codes (describing thread tolerance, thread direction ...) can be added to the above descriptions. For more info on threads and threading, watch:

BENCH WORK PART 4

also

BENCH WORK PART 5

also

THE 1-2-3 BLOCK PROJECT PART 2

also

TAPPING OILS?

also

MORE TAPPING STUFF

also

MACHINE SHOP Q & A PART 1

Also

HOW MACHINE THREADS WORK PART 1

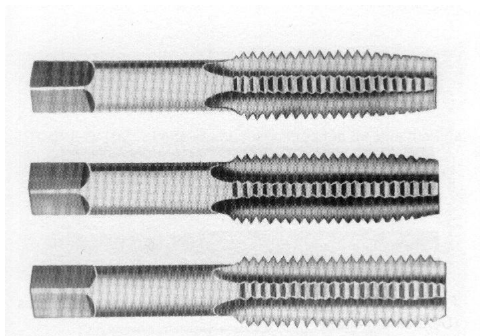
TYPES OF TAPS:

Since there are many types of tapped holes, it is normal to expect that there are several types of taps of a same thread size.

The material being taped, the type of hole, the depth of thread required and the number of holes to tap are the factors that influence the type of tap chosen. The basic types of taps are:

#1: **Straight flute taps** are hand taps and they are the go to taps in the shop. They are less expensive than the other types of taps but they are also less productive. Their straight flutes do not extract the chips produced and this will eventually clog the flutes. That is why these taps must be backed out of the hole regularly in order to clear the chips. In order to be able to tap different types of holes, hand taps are available in three different geometries:

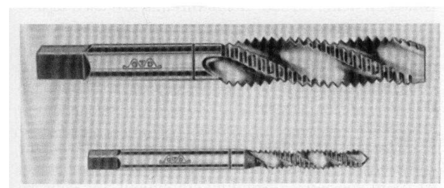
Taper (or starter) taps are used to start a thread in a deep hole or to produce a full thread in a through hole in thin material.



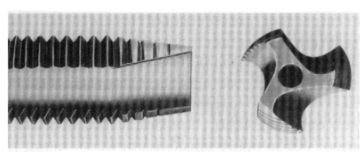
Plug (or intermediate) taps are used to continue a thread started with a taper tap or to produce a full thread in a through hole in a part of medium thickness.

Bottoming (or finishing) taps are used to produce a full complete thread as close as possible to the bottom of a hole.

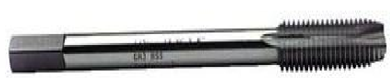
#2: **Helical flute taps are machine taps (not to be used by hand)** that extract the chips produced through the flutes. These production type taps are used to produce threads in blind holes.



#3: **Spiral tip taps** are machine taps that push the chips forward. They are used to produce threads in through holes.



#4: **Relieved shank taps** are used for tapping deep holes. The shank on these taps will not stop the tap from penetrating the part since it is smaller than the minor diameter of the thread. In some cases shank extensions can be used (for tapping very deep holes).



#5: **Forming taps** do not produce chips and do not have flutes, rather they deform the material in order to produce a thread. Formed threads are stronger than cut threads but these taps are complex tools and should only be used by experienced machinists using specialized equipment.

TAP DRILL SIZE:

In order to tap a hole, you must first drill the hole since the tap can only cut the thread. It is important to know that it is impossible cut a full thread using a tap. It is usually recommended (for most materials) to cut only 75% of a full thread (about a 5% reductions in strength). For tough and difficult to tap materials it is possible to reduce thread height to 60% of a full thread. This will have a greater impact on the strength of the thread but hey! IT'S BETTER THAN BRERAKING A TAP!

Only the thread roots are affected by this height reduction since it is the result of a hole that is larger than the minor diameter (root diameter) of the thread.

The formula for calculating the tap drill size for Unified (imperial) threads (75%) is:

nominal diameter - (1 ÷ number of threads per inch)

Example:

1/2" - 13 UNC

Calculation: 0.5" - (1/13)

Answer: 0.423"

The formula for calculating the tap drill size for ISO METRIC threads (75%) is:

nominal diameter - thread pitch

Example: M 10 X 1.5

Calculation: 10 - 1.5

Answer: 8.5 mm

SOME SUGGESTIONS FOR TAPPING:

Taps are very brittle cutting tools. It is very important to avoid breaking the tap in the hole since this could compromise the part being produced but the tap is easy to break so what can you do?

#1: Never use a worn (even slightly) tap or a tap with one or more chipped teeth. Inspect all taps before use, even the new ones!

#2: It is crucial to tap parallel to the axis of the hole.

#3: Use the proper cutting oil for different materials being tapped.

STEEL AND BRONZE: Use a good quality commercial **cutting** oil! Not a **lubricating** oil

CAST IRON: White cast iron cannot be tapped. Grey cast iron is best tapped dry (no liquid or oil). Malleable cast iron is treated the same as mild steel.

ALUMINUM: Light degreasing liquid such as Varsol, kerosene, WD40, a 50/50 mix of dish soap and water or a commercial water based cutting **fluid** (there is a huge difference between a cutting fluid and a cutting oil).

COPPER AND LEAD: Lard, undiluted dish soap, thick way oil, paraffin.

BRASS: Hard brass should be tapped dry, half hard brass should be treated like mild steel and soft brass should be treated like copper.

#4: In order to break and extract the chips when using straight flute taps it is important to back the tap out and clean it regularly.

#5: Since they only cut on one or two teeth, finishing (or bottoming) taps should only be used for finishing the bottom of a blind hole (after having used the starter and plug taps).

<u>TYPES OF THREADING DIES:</u>

Dies are used to cut external threads. There are three main types of threading dies.

SOLID THREADING DIES are not adjustable and are generally used to chase existing or damaged class 2 threads.

SPLIT DIES are slightly adjustable and are used to cut class 1, 2 or 3 threads in readily machinable material. They are also often used to finish threads roughed out on the lathe in tough materials. The slight adjustment permits one roughing and one finishing cut.

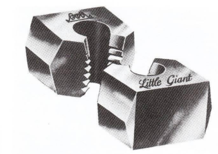


FIGURE B-146 Die halves for two-piece die (TRW, Inc.).

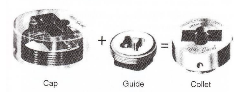


FIGURE B-147 Components of a split adjustable die collet (TRW, Inc.).

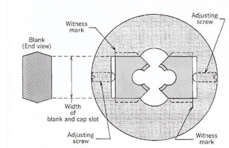


FIGURE B-148 Setting the die position to the witness marks on the die and collet assembly (TRW, Inc.).

TWO PIECE ADJUSTABLE DIES (or adjustable dies) are used for progressive cuts in tough materials or for large (deep) threads. Since the two die halves are adjustable, several roughing passes can be taken, thus reducing the force required for each pass and also reducing the possibility of tearing the thread. They can also be used to rough out threads that will be finished with a solid or split die.

PART DIAMETER FOR DIE THREADING:

Since it is difficult to cut full threads we usually reduce the major diameter of the part being threaded to permit proper crest clearance and avoid seizing the die on the part.

The formula for calculating part diameter for die threading is:

Nominal diameter - (pitch ÷ 7.5)

Metric example:

Thread: **M 10 X 1.5**
Calculation: **10mm - (1.5 ÷ 7.5)**
Answer: **9.8 mm**

Imperial example:

Thread: **1/2 - 13 UNC**
Calculation: **0.5" - ((1/13) ÷ 7.5)**
Answer: **0.489"**

SECTION 5: BLUEPRINT READING

It is crucial that you understand how to read a blueprint. The part being produced will only be acceptable if it conforms to its drawing! Take the time to watch these videos. They aren't exiting, **BUT THEY ARE VERY VERY IMPORTANT!!!**

BLUEPRINT READING PART 1

also

BLUEPRINT READING PART 2

also

BLUEPRINT READING PART 3

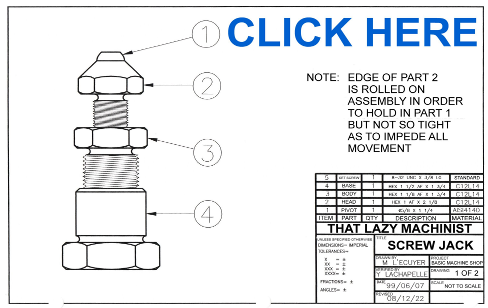
also

BLUEPRINT READING PART 4

also

STANDARD FITS 101, INCH & METRIC

TO DOWNLOAD ALL OF MY PROJECT DRAWINGS



SECTION 6: LAYOUT WORK

6-A: INTRODUCTION

Layout is a preparatory operation during which lines are scribed on the surface of a part in order to accurately define its shape or the limits of a machining operation.

For novice machinists, layout work helps to make the transition from a 2D drawing to a 3D part. For experienced machinists, a layout will speed up the rough machining process and help avoid mistakes.

Example: It is rare for a machinist to miss a dimension by a small amount. An accurate layout gives you something to shoot for. In other words, machine until you get close to the line and then pay attention to the DRO or the graduated collars of the machine.

It is important to layout your part attentively. Check your measurements twice since a good layout could help you avoid an error but a bad layout will inevitably scrap the part.

BENCH WORK PART 6, LAYOUT WORK

ALSO

DRILL POINT GAUGE PART 1

ALSO

DRILL POINT GAUGE'S GRADUATED SCALE

ALSO

THE 1-2-3 BLOCK PROJECT PART 1

ALSO

TAP BLOCK PART 4

ALSO

V-BLOCK PART 6

6-B: BENCHWORK LAYOUTS

Bench work layouts techniques are used to define the limits of a part when it will be produced using low precision tools such as hacksaws, files, drill presses ... .

This layout technique is usually used for parts having a tolerance of +- .010" (0,2 mm) or more. Bench work layouts are rapidly done but require a lot of attention and dexterity.

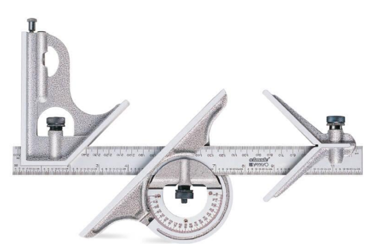
The basic bench work layout tools are:

#1: **Scriber:** Finely pointed tool made of hardened steel or carbide that is used to produce fine lines of the surface of a part.

#2: **Layout blue:** A Prussian blue based die (not paint) applied very thinly on the surface of a clean part before layout in order to accentuate the scribed lines and make them more visible.

To avoid flaking and in order to produce fine and accurate lines it is important to apply only the finest of coats and to do so on a very clean (degreased) surface.

#3: **Adjustable square:** Also known as a



combination set is a very versatile tool used to guide the scriber during layout.

The combination set is made up of four parts. A steel rule on which is mounted the square head (45°and 90°), a protractor head (360° adjustable) and a centre head.

#4: **Dividers:** These adjustable scribers are used to layout arcs and circles from a central point that is usually produced with a prick punch.

NOW WOULD BE A GOOD TIME TO START THE DRILL POINT GAUGE PROJECT!!!

DOWNLOAD THE BLUEPRINTS FOR ALL MY PROJECTS

THEN FOLLOW THE VIDEOS DESCRIBING HOW TO GO ABOUT PRODUCING THE TOOL AS YOU PROGRESS STEP BY STEP THROUGH THE PROJECT

DRILL POINT GAUGE PART 1

ALSO

DRILL POINT GAUGE PART 2

ALSO

DRILL POINT GAUGE PART 3

ALSO

DRILL POINT GAUGE'S GRADUATED SCALE

6-C: PRECISION LAYOUTS

Precision layouts are used to define the limits of a machining operation when the part will be produced on a machine tool other than a drill press.

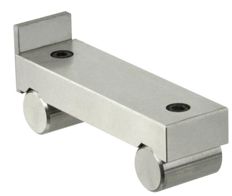
This layout technique is used to produce parts that have a tolerance of +- .003" (0,04 mm) or more.

The most common precision layout tools are:

#1: The **surface plate** is used as a very accurate reference surface from which accurate layout or measurement can be performed.

#2: The **height gauge** is**,** once equipped with a scribe point, used to produce very accurate layout lines by direct measurement since the height gauge has an accurate readable scale (vernier, dial or digital).

#3: The **surface gauge** is, thanks to its accurate base, just as stable as the height gauge but it functions by comparative measurement or by height transfer.



#4: The **sine bar** is a very delicate instrument used with gauge blocks to position parts at an accurate angle to a surface plate.

N.B. The cylindrical supports are very important and must be treated with great care. Never slide a sine bar on a surface plate and when not in use it should be laid on its side.

#5: **Angle plates** are used to position parts vertical to the surface plate. Parts can be mounted on angle plates in order to position them and stabilize them for layout work... and much more.

#6: V-blocks are used to stabilize round or square parts for layout work and much more. They can be used to hold parts horizontally or vertically in relation to the surface plate.

SECTION 7: CHIP PRODUCTION BY PLASTIC DEFORMATION

7-A: INTRODUCTION

Chip production is at the base of most machining operations. If someone wants to understand machining, they must understand the basic principals of chip production. It is this understanding that will guide the machinist when choosing such variables as feed, depth of cut, tool geometry, cutting speed, setup rigidity and choice of cutting fluid.

It is very important to understand that chip production is limited to materials that have a certain amount of resistance to compression and that are malleable. If a material is not malleable, or has very little resistance in compression, it cannot be machined by chip production.

ACCOMPANYING VIDEOS FOR CHIP FORMATION:

CHIPS, SPEEDS AND FEEDS PART 1

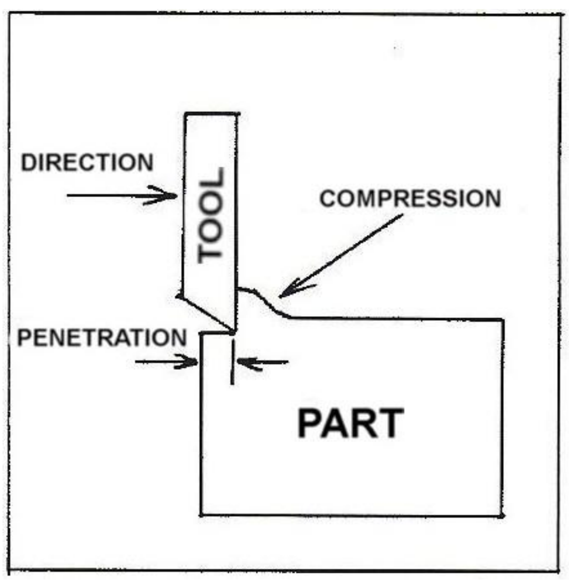
ALSO

CHIPS, SPEEDS AND FEEDS PART 2

ALSO

CUTTING FLUIDS 101

7-B: THE TWO PHASES OF CHIP PRODUCTION

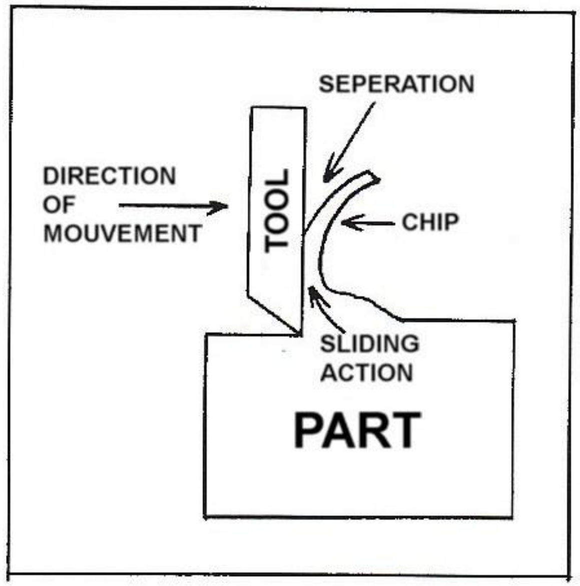


PHASE 1: PENETRATION AND COMPRESSION

We call **penetration** the instant when the tool engages with the part. **Penetration** is immediately followed by the **loading** which is in fact a very violent collision between the tool and the part. Tool fractures often occur during this phase.

The **penetration** and **loading** create an accumulation of compressed material that accumulates just ahead of the tool. The size of this **compression zone** varies proportionately to the part's malleability. The mechanical effort required to deform the material is concentrated as heat in the material accumulated, the more material you compress, the hotter it gets.

PHASE 2: FORMATION AND SEPERATION OF THE CHIP



The chip is **formed** in a way that resembles snow being ploughed. The material accumulated and heated by the tool's progression becomes more fluid and since it has nowhere else to go, it starts to slide up the face of the tool.

The chip **separation** however is the result of a difference of surface tension between both surfaces of the chip. A **properly formed chip** has two surfaces, one that is in **tension** and one that is in **extension**. The curved shape of a properly formed chip is the result of these different forces and it is these forces that separate the chip from the tool.

The **interior surface** of the chip results from the compression of material since it never comes in contact with the tool. It is the least relaxed surface of the chip and that means that it pulls on itself (tension). The interior surface of the chip has a rough and a mat finish and an internally curved surface.

The **exterior surface** of the chip is the result of the flowing (sliding) of the chip up the face of the tool. It is on this surface that is concentrated most of the heat since it has endured the greatest deformation. This intense heat relaxes the tensions (in heat treatment this is called normalizing) caused by the material's compression. The exterior surface of the chip has a clean and shiny appearance and an externally curved surface.

N.B. It is important to know how chips are formed, but it is even more important to understand what variables must be controlled in order to obtain the perfect chip. After all, chip production is a fight between the tool and the part. It is important to control the variables in order to ensure that the tool wins!

7-C: THE VARIABLES

The variables that affect chip formation are: **Cutting speed** (varies with type of material), tool **feed**, **depth of cut**, **tool geometry**, **set up rigidity** and the choice of **cutting fluid**.

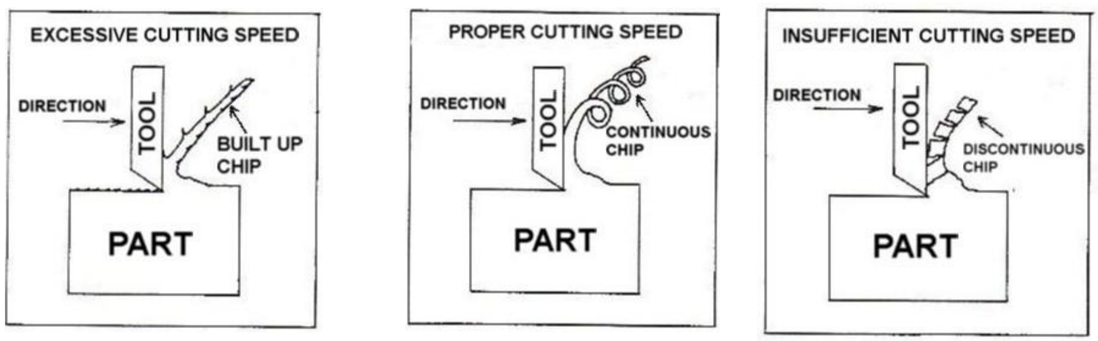
IT IS VERY IMPORTANT TO UNDERSTAND HOW EACH VARIALBLE AFFECTS THE CUTTING ACTION!!!

#1: A material's **cutting speed** is the speed at which the tool, when cutting, will remove a maximum of material (volume) with a minimum of wear (of the tool).

In the **imperial system (inch)** the cutting speed of a material is defined in **feet per minute** and in the **metric system** it is defined as **meters per minute**.

It is important to know that the cutting speeds that the materials manufacturer provides represent a maximum that presupposes that all the other variables are perfectly controlled. It is rare for a home machinist to perform a cut at the maximum speed and it is important to never cut any faster than that speed. Cutting slower than the maximum cutting speed is less productive but it will not melt your tool.

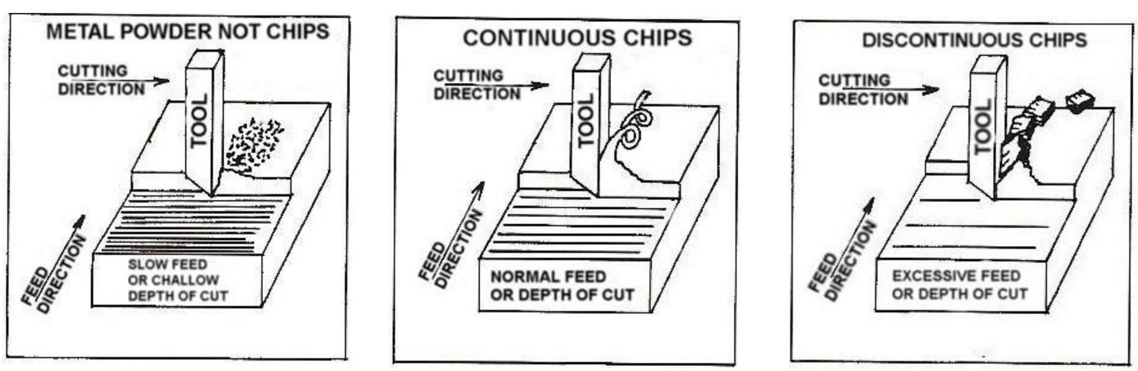
Cutting speed affects chip formation in this way:



#2: **Feed** on the mill is defined as the distance that the table would move in one minute (feet per minute or meters per minute). On the lathe feed is define as the distance that the tool progresses along the part in one turn (inches per revolution or meters per revolution).

Feed determines the width of a chip. For the same cutting speed, a rapid feed will produce a bigger (wider) chip and a slower feed will produce a smaller one.

Feed affects chip formation in this way:

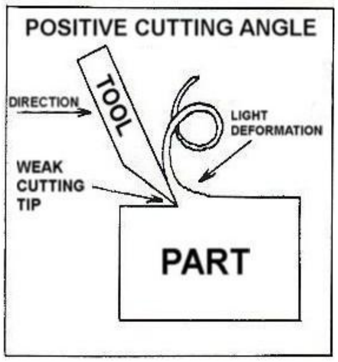


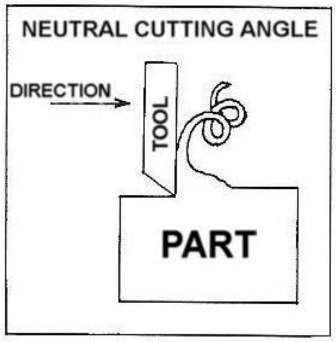
N.B. The feed is the most important variable for surface finish (but not the only one).

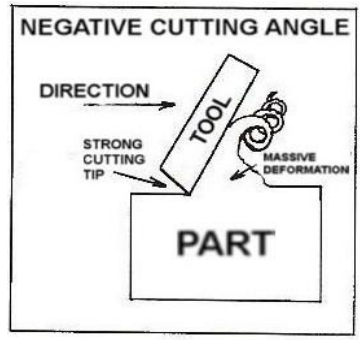
#3: The **depth of cut** has a similar effect on chip formation as the feed. The main difference is that the depth of cut controls the length of cutting lip engagement with the part. Since both variables affect the volume of material removed, it is a good practice to choose the feed first (in order to obtain the proper surface finish) and then adjust the depth of cut for maximum metal removal (roughing) or proper dimension (finishing). Depth of cut has a greater effect on cutting lip engagement comparatively to feed so volume of material removed per minute (productivity) is generally controlled by the depth of cut.

#4: **Tool geometry** affects chip production in multiple ways. For this introductory course we will only consider the effect of the **cutting surface angle** (angle of attack) of the tool.

The **cutting surface angle** affects chip formation in this way:







#5: **The rigidity of the tool**, of the setup, of the machine and of the part being machined are responsible for the amount of deflection during chip production.

There is always some deflection and when well controlled it isn't a problem but an excessive amount of deflection will affect the part's dimensions and geometry as well as the surface finish (since tool deflection is often responsible for vibration).

7-D: CALCULATING THE PROPER R.P.M.

THE VARIABLES:

C.S.: Cutting speed is the optimal linear speed for cutting a given material. This speed is provided by the material's manufacturer or can be found in the MACHINERY'S HANDBOOK. In the imperial system the speed is expressed in **FEET/MINUTE** and metric system speeds are expressed in **METERS/MINUTE**. Since the speed is given as a linear displacement and that most cutting operations use rotary motion, we need a way of changing the linear speed (displacement) to rotary speed (displacement).

DIAMETER: In the formulas that we are using, the diameter is the diameter of what turns during the cut. For drilling and milling it is the diameter of the tool that is used and for turning it is the diameter of the part being turned.

THE FORMULAS:

The formulas for calculating the R.P.M. for high speed steel tools are:

$$\textbf{Imperial:}\quad \frac{CS \times 4}{dia.}$$

$$\textbf{Metric:}\quad \frac{CS \times 300}{dia.}$$

N.B. For carbide cutters, the calculated rpm should multiplied by four.

MOST COMMON CUTTING SPEEDS (CS) USED IN THE HOME SHOP:

		FEET PER MINUTE	METERS PER MINUTE
Steels:	low carbon	**100 Fm**	**30 Mm**
	medium carbon (unhardened):	**70 Fm**	**23 Mm**
	high carbon (unhardened):	**50 Fm**	**16 Mm**
Aluminium:		**300 Fm**	**130 Mm**
Brass:		**200 Fm**	**65 Mm**
Bronze:		**80 Fm**	**26 Mm**

WARNING: THE CALCULATED RPM IS A MAXIMUM. It is preferable to start slower and gradually work your way up to the maximum speed as you adjust your cut . If all the variables are not well controlled you will not cut at maximum speed! Since it is sometimes impossible to fine tune all the variables, it is common to run slower than the maximum RPM.

For examples of cutting speed calculation watch the accompanying

videos suggested at the beginning of this section

NOW WOULD BE A GOOD TIME TO START THE POSITIONNING HAMMER PROJECT!!!

Machine tools required: Drill press, lathe.

HAMMER HANDLE PART 1

ALSO

HAMMER HANDLE PART 2

ALSO

HAMMER HANDLE PART 3

ALSO

4 JAW CHUCK CENTERING

ALSO

HAMMER HEAD

ALSO

HAMMER'S PLUG & TIPS

NOW WOULD BE A GOOD TIME TO START THE 1-2-3 BLOCK PROJECT!!!

Machine tools required: Drill press, vertical milling machine, surface grinder.

THE 1-2-3 BLOCK PROJECT PART 1

ALSO

THE 1-2-3 BLOCK PROJECT PART 2

ALSO

THE 1-2-3 BLOCK PROJECT PART 3

SECTION 8: MATERIALS AND METALLURGY

For most beginners, what counts is cutting. We took some time to look at blueprint reading because if we can't define what we have to attain as far as precision and form goes, we can't produce the part. The same logic applies to materials. Understanding what we are cutting is very important if you wish to produce accurate parts that have proper surface finishes. Here are some videos that will help you anticipate what you are up against when machining most materials. Many hobby machinists don't spend a lot of time thinking about the materials they are cutting, and that will cause problems. Remember that any project starts with a chunk of material. What that chunk is will define what you can do to it.

YOU ARE NOT THE BOSS! THE MATERIAL IS!!!

MATERIALS PART 1

ALSO

MATERIALS PART 2

ALSO

MATERIALS PART 3

ALSO

MATERIALS PART 4

ALSO

INTRODUCTION TO FERROUS METALS

ALSO

RECOGNIZING STEEL ALLOYS, HOME TESTS &TRICKS

ALSO

MATERIALS, TESTING 101

ALSO

HEAT TREATMENT OF STEELS PART 1

ALSO

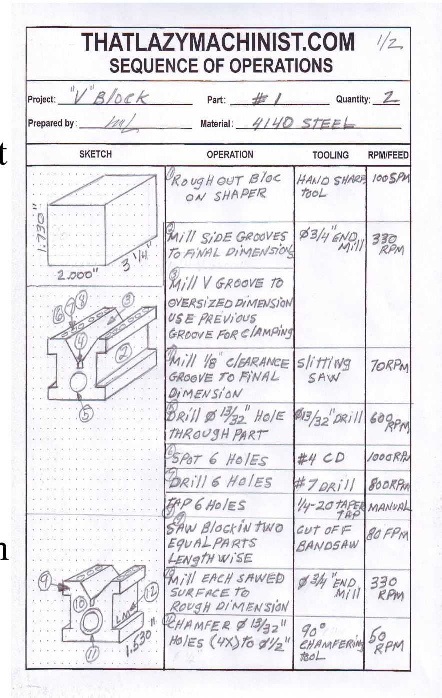
HEAT TREATMENT OF STEELS PART 2

ALSO

INJECTION MOLDING 101

SECTION 9: SEQUENCE OF OPERATIONS

You may not want to read this but planning the sequence of operations to perform on a part is almost as important as the machining itself! And it is way more boring. It is not rare for novice machinists to cut themselves into a corner. In other words, performing certain operations before others is crucial in order to ensure that there is material left to hold, that the hole is not too large for the thread, that you have and maintain proper reference surfaces all through the parts productions, that.....



THATLAZYMACHINIST.COM 1/2
SEQUENCE OF OPERATIONS

Project: "V" Block Part: #1 Quantity: 2
Prepared by: ML Material: 4140 STEEL

SKETCH	OPERATION	TOOLING	RPM/FEED
	1 Rough out bloc on shaper	Hand sharp. tool	100SPM
	2 Mill side grooves to final dimensions	⌀3/4" end mill	330 RPM
	3 Mill V groove to oversized dimension use previous groove for clamping		
	4 Mill 1/8" clearance groove to final dimension	Slitting saw	70RPM
	5 Drill ⌀ 13/32" hole through part	⌀13/32" drill	600 RPM
	6 Spot 6 holes	#4 CD	1000RPM
	7 Drill 6 holes	#7 drill	800RPM
	8 Tap 6 holes	1/4-20 taper tap	Manual
	9 Saw block in two equal parts lengthwise	Cut off bandsaw	80 FPM
	10 Mill each sawed surface to rough dimension	⌀3/4" end mill	330 RPM
	11 Chamfer ⌀ 13/32" holes (4X) to ⌀1/2"	90° chamfering tool	50 RPM

When you watch one of my project videos, you may notice that nothing is done for no reason and that I am following a clear and well though out path that will maximize my chances for success. Here is how you can go about planning a sequence of operations.

[*SEQUENCE OF OPERATIONS PART 1*]

ALSO

[*SEQUENCE OF OPERATIONS PART 2*]

ALSO

[*SEQUENCE OF OPERATIONS PART 3*]

ALSO

[*SEQUENCE OF OPERATIONS PART 4*]

ALSO

[*V-BLOCK PART 1*]

ALSO

[*SCREW JACK PART 1*]

SECTION 10: MACHINE TOOLS & MACHINING

In this section we will look at different machine tools and how to use them. This part of the online course will be covered by project and theory videos. So have fun and watch away but pace yourself and be wary of video overdose. This is by far the longest section so remember, the objective is to learn (not to complete as fast as possible) so take your time and don't hesitate to watch the videos more than once. Also take the time to look at the comments, they often contain viewer input or answers for common questions. P.S. Some videos appear in more than one type of machine tool video list since most projects require the use of multiple machine tools to complete.

10-A: DRILL PRESS

THE DRILL PRESS PART 1

ALSO

THE DRILL PRESS PART 2

ALSO

THE DRILL PRESS PART 3

ALSO

THE DRILL PRESS PART 4

ALSO

THE DRILL PRESS PART 5

ALSO

DRILL BITS FOR METAL , HOW THEY WORK!

ALSO

HOW TO SHARPEN DRILLS, OLD SCHOOL

ALSO

DRILL POINT GAUGE PART 1

ALSO

10-B: LATHE

ALSO

HAMMER PLUG & TIPS

ALSO

PROJECTING HARD TO GET TO SURFACES

ALSO

THREADING ON THE LATHE

ALSO

THREAD CHASING DIALS
& NON STANDARD THREADS

ALSO

HOW MACHINE THREADS WORK

ALSO

EFFECT OF PLUNGE ANGLE ON THREAD CUTTING

ALSO

TURNING A HALF BALL!

ALSO

MACHINING A SPHERE!

ALSO

SCREW JACK PART 1

ALSO

SCREW JACK PART 2

ALSO

SCREW JACK PART 3

ALSO

SCREW JACK PART 4

ALSO

SCREW JACK PART 5

10-C: MILLING MACHINE

ALSO

SPUR GEARS & DIVIDING HEADS

ALSO

POCKET MILLING BY COORDINATES

ALSO

ROUND COLUMN MILL PART 1

ALSO

ROUND COLUMN MILL PART 2

ALSO

ROUND COLUMN MILL MODIFICATIONS

ALSO

V-BLOCK PART 4

ALSO

V-BLOCK PART 5

ALSO

V-BLOCK PART 6

ALSO

V-BLOCK PART 7

ALSO

TAP BLOCK PART 1

ALSO

TAP BLOCK PART 2

ALSO

TAP BLOCK PART 3

10-D: SHAPER

10-E: GRINDERS

[TAP BLOCK PART 6]

ALSO

[THE 1-2-3 BLOCK PROJECT PART 3]

ALSO

CYLINDRICAL GRINDING 101

10-F: BAND SAWS

[V-BLOCK PART 7]

ALSO

[BAND SAW BLADE WELDING]

I hope you had fun and that these notes and the accompanying videos were helpful. If you wish to, you can download (AVAILABLE IN NOVEMBER 2018) the test sheets from my web page:

THATLAZYMACHINIST.COM

and return your completed sheets to me at

THATLAZYMACHINIST@GMAIL.COM

and I will send you (after evaluation) a personalized certificate for having completed the theory portion of this online course.

Marc L'Ecuyer

THAT LAZY MACHINIST