can be used on soft, easily penetrated materials than on hard materials which naturally tend to dull the wheel faster. The softer grades release the dull grains more readily to present new, sharp grains to the work.

2. Second factor, in selecting a wheel in the amount of stock to be removed and the finish required. These affect the choice of grift size and bond as follows:

- A relatively coarse grit size is selected for rapid stock removal without regard for finish as rough grinding; a fine grift should be used where a high finish is desired.
- Vitrified bonded wheels are generally used where a commercial finish satisfactory. The organic bonds, resinoid, rubber and shellac, produce the highest finish.

3. The area of grinding contact between the wheel and the work affects the choice of grift size and grade.

- A coarse grift is required when the contact area is relatively large, as in surface grinding with cup wheels, cylinders or segments, to provide adequate chip clearance between the abrasive grains. As area of contact becomes smaller and the unite pressure tending to break down the wheel face becomes greater, finer grit wheels should be used.
- As to the grade or hardness, on large area of contact a soft grade will provide normal breakdown of the wheel, insuring continuous, free-cutting action. A harder grade, on the other hand, is needed to stand up under the increasingly higher unit pressure as the area of contact becomes smaller.

4. The severity of the grinding operation affects the choice of abrasive and grade.

- A tough abrasive like 4A Aluminum Oxide should be used for rough, heavy duty grinding of steel.
- The milder abrasives like 32 and 38 Aluminum Oxide are best for lighter precision grinding operations on steels and semisteels, while the intermediate 57 and 19 Aluminum Oxide abrasives are used for precision and semiprecision grinding of both mild and hard steels.
- The severity of the grinding operation also influences the choice of grade. Hard grade provide durable wheels for rough grinding such as snagging, while medium and softer grade wheels can be used for precision type operations which are less severe on the wheel.

5. The speed at which the grinding wheel is to be operated often dictates the type of bond.

- Vitrified bonded wheels should not be used at speeds over 6,500 s.f.p.m. With few exceptions, when the speed exceeds this figure, resinoid, rubber or shellac bonded wheels should be used. Note, the safe operating speed shown on the tag, wheel or blotter must never be exceeded.

6. Feed rate

- The higher the feed rate, the greater the grinding pressure is. If the grinding speed of workpiece must be increased, the feed rate will be increased, then the wear of the wheel will be faster. Therefore a harder grinding wheel is required.

- A standard wheel marking system is used for the identifying five major factors in grinding wheel selection:
  - Type of abrasive
  - Grit size
  - Grade or hardness
  - Structure
  - Bond

First Symbol: Type of Abrasive

A wheel marked A 60-J8V indicates the following:

A – Fused aluminum oxide

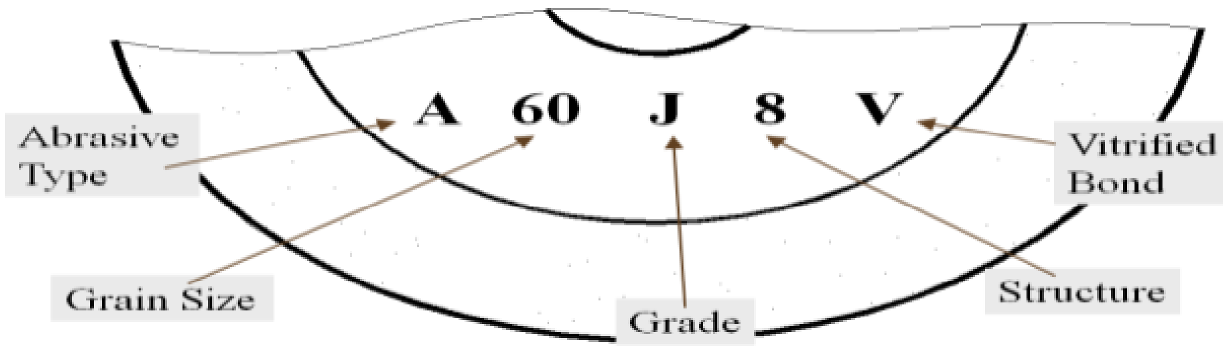
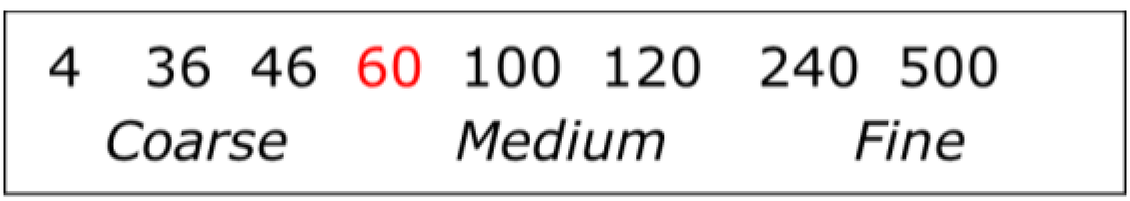


Figure 8: Grinding Wheel Marking

Second Symbol: Grit Size

The following scale can be used to determine grit:



Third Symbol: Grade of Hardness

- Hardness grade is a measure of bond strength of the grinding wheel.
  - Bond material holds abrasive grains together in the wheel.
  - The stronger the bond, the harder the wheel.
- Hardness grade is a measure of bond strength of the grinding wheel.

A to G are softer.
H to P are more medium grades.
R to Z are harder.

Fourth Symbol: Structure

- Structure, the spacing of the abrasive grains in the wheel is indicated by numbers.

1 is a dense structure.
8 is a more medium structure.
15 is an open structure.

Fifth Symbol: Bond

- Bond is identified by letter according to the following:
  - V – Vitrified
  - B – Resinoid

- R – Rubber
- E – Shellac
- M – Metal

Standard grinding wheel marking example:

1- A – 305 X 25 X 127 WA 46 K 8 V 7N 2000m/min

FROM(WHEEL TYPE): 1(Straight-plain)

FACE: A

SIZE: Dia. (D) X Width(W) X Bore(H)

ABRASIVE TYPE: WA (See Figure 2)

GRAIN SIZE: 46 (See Figure 2)

GRADE: K (See Figure 2)

STRUCTURE: 8 (See Figure 2)

BONE TYPE: V (See Figure 2)

MAKER CODE: 7N

MAX. RPM: 2000m/min.

WA		60	K	7	V
ABRASIVES		GRIT SIZE	GRADE	STRUCTURE	BOND TYPE
A	Regular Aluminium Oxide	10 Coarse	A Soft	1 Dense	V : Vitrified
WA	White Aluminium Oxide	12	B	2	B : Resinoid
19A	Mixture of A&WA	14	C	3	R : Rubber
FA	Semi-friable Aluminium Oxide	16	D	4	O : MgO
PA,RA	Pink Aluminium Oxide	20	E	5	E : Epoxy
SA(HA)	Single Crystal Aluminium Oxide	24	F	6	
23A	Mixtue of A$SA	30	G	7 To	
AZ	Zirconium Oxide	36	H	8	
C	Black Silicon Carbide	46	I	9	
GC	Green Silicon Carbide	54	J	10	
RC	Mixture of C&GC	60 To	K To	11	
		80	L	12	
		100	M	13	
		120	N	14 Open	
		150	O		
		180	P		
		220	Q		
		280	R		
		320	S		
		400	T		
		500	U		
		600	V		
		800	W		
		1000	X		
		1200 Fine	Y		
			Z Hard		

Figure 9: Grinding Wheel Selection Chart

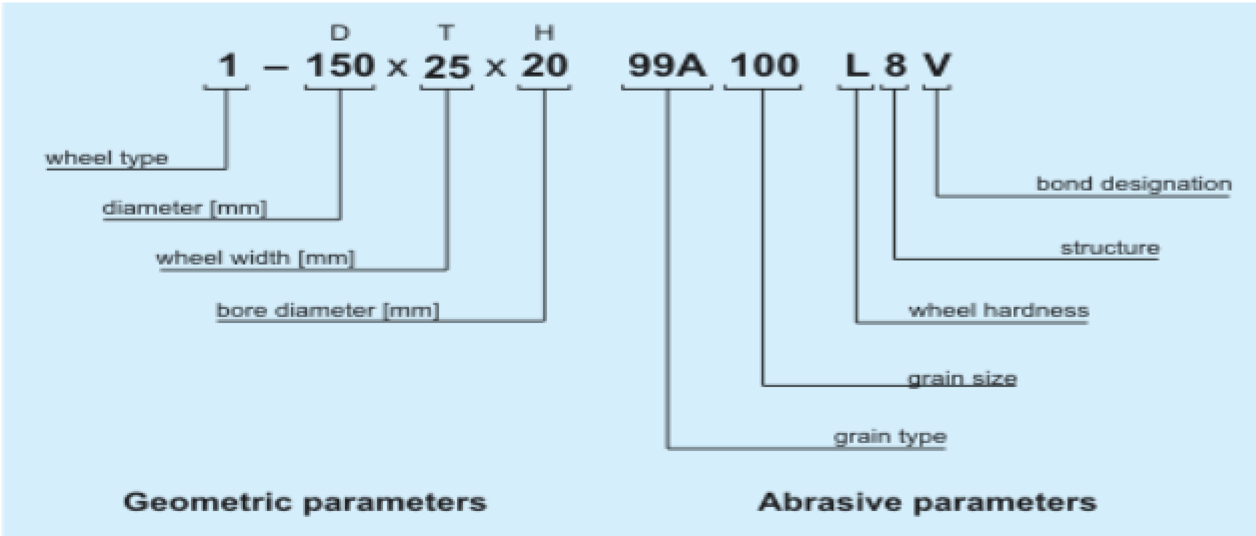


Figure 10: Grinding Wheel Selection Chart

UNITS TEST

1. Please list five Safety Precautions.

2. Please list five main parts of the surface grinders.

3. What is a diamond wheel dresser?

4. When Dressing the Wheel how far Diamond dresser should be located to the left of the center of the wheel?

5. What is a Ring Test?

6. How do you Performing the ring test?

7. When select the grinding wheel, there are eight factors which affect the choice of the grinding wheel specifications. Please list five out of eight factors.

8. Aluminum oxide grinding wheel are best for what?

9. A standard wheel marking system is used for the identifying factors in grinding wheel selection. Please all five major factors?

10. A wheel marked WA 80-L9B, Please indicates the following.

Chapter Attribution Information

This chapter was derived from the following sources.

Grinding and Buffing derived from Mechanical Engineering Tools by the Massachusetts Institute of Technology,

PART VI

Chapter 6: Heat Treating

Chapter 6: Heat Treating

OBJECTIVE

After completing this unit, you should be able to:

- Correctly harden a piece of tool steel and evaluate your work.
- Correctly temper the hardened piece of the tool steel and evaluate your work.
- Describe the proper heat treating procedures for other tool steels.

Safety

The following procedures are suggested for a safe heat treating operation.

1. Wear heat-resistant protective clothing, gloves, safety glasses, and a face shield to prevent exposure to hot oils, which can burn skin.
2. Before lighting the furnace, make sure that air switches, exhaust fans, automatic shut-off valves, and other safety precautions are in place.
3. Make sure that there is enough coolant for the job. Coolant will absorb heat given off by the metal as it is cooling, but if there is insufficient coolant, the metal will not cool at the optimal speed.
4. Make sure that there is sufficient ventilation in the quenching areas in order to maintain desired oil mist levels.
5. When lighting the furnace, obey the instructions that have been provided by the manufacturer.
6. During the process of lighting an oil or gas-fired furnace, do NOT stand directly in front of it.
7. Make sure that the quenching oil is not contaminated by water. Explosions can be results of moisture coming into contact with the quenching oil.
8. Before taking materials out of the liquid carburizing pot, make sure that the tongs are not wet and that they are the correct tongs for the job.
9. Make sure that an appropriate fungicide or bacterial inhibitor has been mixed into the quenching liquid.
10. When quench tanks are not being used, always cover them.
11. Use a nonflammable absorbent to clean leaks and oil spills. This should be done immediately.
12. If possible, keep tools, baskets, jigs, and work areas free from oil contamination.
13. Before breaks and before moving on to the next task, wash your hands thoroughly.
14. If any skin trouble is shown or suspected, report to your instructor and get medical help.
15. Fumes from the molten carburizing salt bath should not be inhaled, because carbon monoxide is a product of the carburizing process.
16. Make sure there is good ventilation in the work area.
17. Be on the lookout for contamination from pieces of carburized metal.
18. Do not take oil-soaked clothes or equipment to areas where there are food or beverages.
19. Do not take food or beverages where oils are either being used or stored.

Procedure

The first important thing to know when heat treating a steel is its hardening temperature. Many steels, especially the common tool steels, have a well established temperature range for hardening. O-1 happens to have a hardening temperature of 1450 – 1500 degrees Fahrenheit.

To begin the process:

1. Safety first. Heat treating temperatures are very hot. Dress properly for the job and keep the area around the furnace clean so that there is no risk of slipping or stumbling. Also, preheat the tongs before grasping the heated sample part.

2. Preheat the furnace to 1200 degrees Fahrenheit.

3. When the furnace has reached 1200 degrees Fahrenheit, place the sample part into the furnace. Place the sample part into the center of the oven to help ensure even heating. Close and wait.

4. Once the sample part is placed in the furnace, heat it to 1500 degrees Fahrenheit. Upon reaching this temperature, immediately begin timing the soak for 15 minutes to an hour (soak times will very depending on steel thickness).

Table 1: Approximate Soaking Time for Hardening, Annealing and Normalizing Steel

Thickness Of Metal (inches)	Time of heating to required Temperature (hr)	Soaking time (hr)
up to 1/8	.06 to .12	.12 to .25
1/8 to 1/4	.12 to .25	.12 to .25
1/4 to 1/2	.25 to .50	.25 to .50
1/2 to 3/4	.50 to .75	.25 to .50
3/4 to 1	.75 to 1.25	.50 to .75
1 to 2	1.25 to 1.75	.50 to .75
2 to 3	1.75 to 2.25	.75to 1.0
3 to 4	2.25to 2.75	1 to 1.25
4 to 5	2.75 to 3.50	1 to 1.25
5 to 8	3.50 to 3.75	1 to 1.50

Soak time is the amount of time the steel is held at the desired temperature, which is in this case 1500 degrees Fahrenheit.

5. When the soak time is complete, very quickly but carefully take the sample out with tongs. Place the sample part into a tank of oil for quenching. Move the sample part around as much as possible while it is quenching.

6. Once the sample part has been quenched down to around 125 degrees Fahrenheit, begin the tempering process. To temper the sample part it must be placed into the furnace at 375 degrees Fahrenheit. Allow it to soak for 2 hours, then remove the sample part and allow it to cool to room temperature. The sample part should now be approximately at a hardness of 60 RC.

Austenitize and Air-Cool:

1. This heat treatment is usually done by the manufacturer, which results in it being called the as-received condition

2. To reach this state, a process called normalizing (also called the thermal history) is done. Normalizing 1045 steel usually consists of these steps:

    2.1 Austenitize: Place the steel in the furnace at 1562°F in the austenite range, and keep it there for an

hour until the metal has reached its equilibrium temperature and corresponding solid solution structure.
2.2 Air-cool: Take the steel out of the furnace and let it air-cool to room temperature.

Austenitize and Furnace-Cool (Annealing):

1. This process is also referred to as annealing. During annealing, the steel goes through the following temperature histories:
    1.1 Austenitize: Place the steel in the furnace at 1562°F in the austenite range, and keep it there for an hour until the metal has reached its equilibrium temperature and corresponding solid solution structure.
    1.2 Furnace-Cool: cool the steel slowly in the furnace. Allow the temperature to drop from 1562°F to 1292°F over a ten hour period.
    1.3 Air-cool: Take the steel out of the furnace and let it air-cool to room temperature.

Austenitize and Quench:

1. Austenitize: Place the steel in the furnace at 1562°F in the austenite range, and keep it there for an hour until the metal has reached its equilibrium temperature and corresponding solid solution structure.
2. Quench: quickly remove the steel from the furnace, plunge it into a large container of water at room temperature, and stir vigorously. When using 1045 steel, room temperature water is used as the quenching medium.

Quench: Rapidly remove material from furnace, plunge it into a large reservoir of water at ambient temperature, and **stir vigorously.**

For 1045 steel, the quenching medium is water at room temperature (for other steels, other quenching media such as oil or brine are used).

4. Austenitize, Quench, and Temper:

1. Austenitize: Place the steel in the furnace at 1562°F in the austenite range, and keep it there for an hour until the metal has reached its equilibrium temperature.
2. Quench: Quickly remove the steel from the furnace, plunge it into a large container of water at room temperature, and stir vigorously.
3. Temper:
    3.1 Bring the steel to the tempering temperature and hold it there for about 2 hours.
    3.2 There is a range of different tempering temperatures. For 1045 steel the range is from 392 to 932°F.
    3.3 The different temperatures lead to differences in mechanical properties.
    3.4 Lower temperatures give higher yield strength but lower toughness and ductility.
    3.5 Higher temperatures give lower strength but increase toughness and ductility.
4. Air-cool: Take the steel out of the furnace and let it air-cool to room temperature.

16

Unit 2: Hardness Testing

OBJECTIVE

After completing this unit, you should be able to:

- Perform a Rockwell Test
- Perform a Brinell Test

Beyond verifying our in shop heat treatment, testing hardness is sometime necessary for production work as well. Even though it's bad planning, occasionally a job arrives at our machine shop with an unknown alloy or maybe its composition is known but the hardness isn't. It is possible to use a file to roughly test the machinability of that metal, but the best way to select cutter types, speed, and feeds is a true hardness measurement.

Brinell: Testing hardness by reading the diameter of a ball penetrator mark.

Rockwell: Testing hardness by reading a penetrator depth.

The Rockwell Hardness Test

The Rockwell is a widely accepted method for both soft and hard metals. This system gauges malleability by measuring the depth a pointed probe known shape and size will penetrate into the material given an exact amount of force upon it. Due to Rockwell's range it is the most popular test in tooling and small production shops and training labs.

Rockwell Numbers:

There are several different scales within the Rockwell system. We'll use the Rockwell C scale, correctly used on the hardened steel. The C scale can be said to start at 0(annealed Steel) and run up to 68, harder than a HSS tool bit, near that of a carbide tool. It is symbolized by a larger R with the scale subscript.

$$R_C$$

The two step Rockwell test:

Step 1. Calibrate Load

The test object is set upon the lower anvil such that it's stable and won't move when pressed down from above. Next, a cone-shaped diamond penetrator is brought into contact then driven into metal to a predetermined of 20lbs. That cause the conical point to sinks into the metal from 0.003 to 0.006 inch. This is the initial calibration load. At that time, a large dial indicator is rotated to read zero.

Step 2.

Test Load Then with the calibration pressure upon the penetrator and indicator set to zero, a second addition 20lbs test load is added. As the diamond sink farther, its added depth is translated to dial, but in an inverse relationship. The deeper the diamond penetrates, the softer the metal tests, therefore the lower number that must appear on the dial face. Inversely, when the point can't go very deep, the metal is hard and registers higher on the dial face.

The Rockwell Method

The Rockwell method measures the permanent depth of indentation produced by a force/load on an indenter.

1. Prepare the sample.

2. Place the test sample on the anvil.
3. A preliminary test force (commonly referred to as preload or minor load) is applied to a sample using a diamond indenter.
4. This load represents the zero or reference position that breaks through the surface to reduce the effects of surface finish. After the preload, an additional load, called the major load, is applied to reach the total required test load.
5. This force is held for a predetermined amount of time (dwell time: 10-15 seconds) to allow for elastic recovery.
6. This major load is then released and the final position is measured against the position derived from the preload, the indentation depth variance between the preload value, and the major load value. This distance is converted to a hardness number.

The Brinell Hardness Test

The Brinell Hardness test is very similar to the Rockwell system in that a penetrator is forced into the sample, however, here the measured gauge is the diameter of the dent made by penetration of a hard steel ball of known size, into the work-piece surface. Hardened tool steel balls are used for testing softer material, while a carbide penetrator ball is used to test harder metals.

Due to the upper hardness, limiting factor of Brinell ball, this test it correctly used as a test of soft to medium hard metals.

The Brinell scale numbers:

The scale runs from 160 for annealed steel up to approximately 700 for very hard steel.

The Brinell hardness test is an alternative way to test the hardness of metals and alloys.

1. Prepare the sample.
2. Place the test sample on the anvil.
3. Move the indenter down into position on the part surface.
4. A minor load is applied and a zero reference position is established.
5. The major load is applied for a specified time period (10 to 15 seconds) beyond zero.
6. The major load is released, leaving the minor load applied.
7. Follow the process to determine the Brinell hardness of an aluminum sample.
    - 7.1 Press the indenter into the sample using an accurately controlled test force.
    - 7.2 Maintain the force for a specific dwell time (usually 10 to 15 seconds).
    - 7.3 After the dwell time is complete, remove the indenter, leaving a round indent in the sample.
    - 7.4 The size of the indent is determined optically by measuring two diagonals of the round indent using either a portable microscope or one that is integrated with the load application device.
    - 7.5 The Brinell hardness number is a function of the test force divided by the curved surface area of the indent. The indentation is considered to be spherical, with a radius equal to half the diameter of the ball. The average of the two diagonals is used in the following formula to calculate the Brinell hardness.

BHN=F2D(D-D2- d2)

Unit Test:

1. Please lists five heat treating safety.
2. What is first important thing to know when heat treating a steel?
3. What is the soak time for 1 to 2” Thickness Of Metal?
4. Please explain Soak time.
5. After the soak time is complete, what is the next step?
6. To temper the sample part it must be placed into the furnace at what temperature?
7. Please explain Austenitize and Quench.
8. What is a Air-cool?
9. Please explain The Rockwell Method.
10. Please explain The Brinell Hardness Test.

Chapter Attribution Information

This chapter was derived from the following sources.

- **Heat Treating** derived from Heat Treatment of Plain Carbon and LowAlloy Steels by the Massachusetts Institute of Technology, CC:BY-NC-SA 4.0.
- **Brinell Hardness Test Equation** derived from Brinell Hardness by CORE-Materials Resource Finder, CC:BY.

PART VII

Chapter 7: Lean Manufacturing

Chapter 7: Lean Manufacturing

OBJECTIVE

After completing this unit, you should be able to:

- Apply 5S in any Machine shop.
- Describe Kaizen Concept.
- Describe Implementing Lean Manufacturing.

Lean 5S:

"5S" is a method of workplace organization that consists of five words: Sort, Set in order, Shine, Standardize, and Sustain. All of these words begin with the letter S. These five components describe how to store items and maintain the new order. When making decisions, employees discuss standardization, which will make the work process clear among the workers. By doing this, each employee will feel ownership of the process.

Phase 0: Safety

It is often assumed that a properly executed 5S program will improve workplace safety, but this is false. Safety is not an option; it's a priority.

Phase 1: Sort

Review all items in the workplace, keeping only what is needed.

Phase 2: Straighten

Everything should have a place and be in place. Items should be divided and labeled. Everything should be arranged thoughtfully. Employees should not have to bend over repetitively. Place equipment near where it is used. This step is a part of why lean 5s is not considered "standardized cleanup".

Phase 3: Shine

Make sure that the workplace is clean and neat. By doing this, it will be easier to be aware of where things are and where they should be. After working, clean the workspace and return everything to its former position. Keeping the workplace clean should be integrated into the daily routine.

Phase 4: Standardize

Standardize work procedures and make them consistent. Every worker should be aware of what their responsibilities are when following the first three steps.

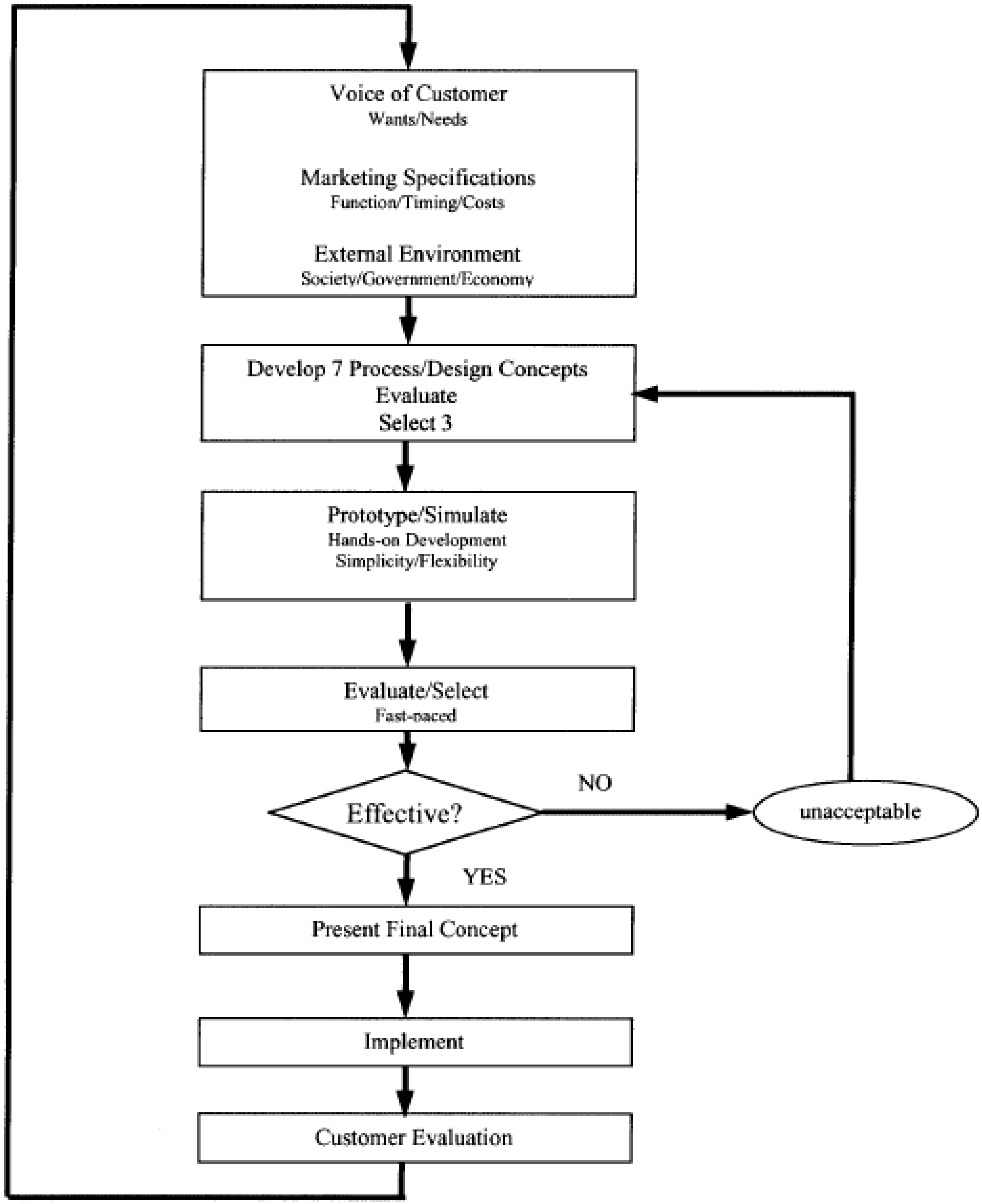
Phase 5: Sustain

Assess and maintain the standards. The aforementioned steps should become the new norm in operation. Do not gradually revert to the old ways. When taking part of the new procedure, think of ways to improve. Review the first four steps when new tools or output requirements are presented.

Kaizen

While the lean 5S process focuses on the removal of waste, Kaizen focuses on the practice of continuous improvement. Like lean 5S, Kaizen identifies three main aspects of the workplace: **M**uda (wastes), **M**ura (inconsistencies) and **M**uri (strain on people & machines). However, the Kaizen step-by-step process is more extensive that the lean 5S process.

The Kaizen process overview:



1. Identify a problem.
2. Form a team.
3. Gather information from internal and external customers, and determine goals for the project.
4. Review the current situation or process.
5. Brainstorm and consider seven possible alternatives.
6. Decide the three best alternatives of the seven.
7. Simulate and evaluate these alternatives before implementation.
8. Present the idea and suggestions to managers.
9. Physically implement the Kaizen results and take account of the effects.

Lean manufacturing improves as time goes one, so it is important to continue education about maintaining standards. It is crucial to change the standards and train workers when presented with new equipment or rules.

Lean

Think of a maintenance department as serving internal customers: the various departments and workers in the company.

Lean is different from the traditional western, mass production model that relies on economies of scale to create profits. The more you make the cheaper the product will become, the greater the potential profit margin. It is based on predictions of customer needs, or creating customer needs. It has difficulty dealing with unusual changes in demand.

Lean production responds to proven customer demand. Pull processing – the customer pulls production. In a mass system the producer pushes product onto the market, push processing.

Building a long-term culture that focuses on improvement.

Respect for workers better trained and educated, more flexible

Lean is a philosophy that focuses on the following:

1. Meeting customer needs
2. Continuous, gradual improvement
3. Making continuously better products
4. Valuing the input of workers
5. Taking the long term view
6. Eliminating mistakes
7. Eliminating waste

Wastes: using too many resources (materials, time, energy, space, money, human resources, poor instructions)

Wastes:

1. Overproduction
2. Defects
3. Unnecessary processing
4. Waiting (wasting time)
5. Wasting human time and talent
6. Too many steps or moving aroundExcessive transportation
7. Excessive inventory

Lean production includes working with suppliers, sub contractors, and sellers to stream line the whole process.

The goal is that production would flow smoothly avoiding costly starts and stops.

The idea is called just in time "produce only what is needed, when it is needed, and only in the quantity needed." Production process must be flexible and fast.

Inventory = just what you need

In mass production = just in case. Extra supplies and products are stored just in case they are needed.

Terminology:

Process simplification – a process outside of the flow of production

Defects – the mass production system does inspection at the end of production to catch defects before they are shipped. The problem is that the resources have already been "spent" to make the waste product" Try to prevent problems immediately, as they happen, then prevent them. Inspection during production, at each stage of production.

Safety – hurt time is waste time

Information – need the right information at the right time (too much, too little, too late)

Principles:

Poka-yoke – mistake proof determining the cause of problems and then removing the cause to prevent further errors

Judgment errors – finding problems after the process

Informative inspections – analyzing data from inspections during the process

Source inspections – inspection before the process begins to prevent errors.

MEAN LEAN

One of the terms applied to a simply cost cutting, job cutting interpretation of Lean is Mean Lean. Often modern manager think they are doing lean without understanding the importance of workers and long term relationships.

Reliability Centered Maintenance

Reliablity centered maintenance is a system for designing a cost effective maintenance program. It can be a detailed complex, computer, statistically driven, but at its basics it is fairly simple. Its ideas can be applied to designing and operating a PM system, and can also guide your learning as you do maintenance, troubleshooting, repair and energy work.

These are core principles of RCM. These nine fundamental concepts are:

- Failures happen.
- Not all failures have the same probability
- Not all failures have the same consequences
- Simple components wear out, complex systems break down
- Good maintenance provides required functionality for lowest practicable cost
- Maintenance can only achieve inherent design reliability of the equipment
- Unnecessary maintenance takes resources away from necessary maintenance
- Good maintenance programs undergo continuous improvement.

Maintenance consists of all actions taken to ensure that components, equipment, and systems provide their intended functions when required.

An RCM system is based on answering the following questions:

1. What are the functions and desired standards of performance of the equipment?
2. In what ways can it fail to fulfil its functions? (Which are the most likely failures? How likely is each type of failure? Will the failures be obvious? Can it be a partial failure?)
3. What causes each failure?
4. What happens when each failure occurs? (What is the risk, danger etc.?)
5. In what way does each failure matter? What are the consequences of a full or partial failure?
6. What can be done to predict or prevent each failure? What will it cost to predict or prevent each failure?
7. What should be done if a suitable proactive task cannot be found (default actions) (no task might be available, or it might be too costly for the risk)?

Equipment is studied in the context of where when and how it is being used

All maintenance actions can be classified into one of the following categories:

- Corrective Maintenance – Restore lost or degraded function
- Preventive Maintenance – Minimizes opportunity for function to fail
- Alterative Maintenance – Eliminate unsatisfactory condition by changing system design or use

Within the category of preventive maintenance all tasks accomplished can be described as belonging to one of five (5) major task types:

- Condition Directed – Renew life based on measured condition compared to a standard
- Time Directed – Renew life regardless of condition
- Failure Finding – Determine whether failure has occurred
- Servicing – Add/replenish consumables
- Lubrication – Oil, grease or otherwise lubricate

We do maintenance because we believe that hardware reliability degrades with age, but that we can do something to restore or maintain the original reliability that pays for itself.

RCM is reliability-centered. Its objective is to maintain the inherent reliability of the system or equipment design, recognizing that changes in inherent reliability may be achieved only through design changes. We must understand that the equipment or system must be studied in the situation in which it is working.

Implementing Lean Manufacturing

Analyze each step in the original process before making change

Lean manufacturing main focuses is on cost reduction and increases in turnover and eliminating activities that do not add value to the manufacturing process. Basically what lean manufacturing does is help companies to achieve targeted production, as well as other things, by introducing tools and techniques that are easy to apply and maintain. What these tools and techniques are doing is reducing and eliminating waste, things that are not needed in the manufacturing process.

Manufacturing engineers set out to use the six-sigma DMAIC (Design, Measure, Analyze, Improve, Control) methodology—in conjunction with lean manufacturing—to meet customer requirements related to the production of tubes.

Manufacturing engineers were charged with designing a new process layout of the tube production line. The objectives for project were including:

- Improved quality
- Decreased scrap
- Delivery to the point of use
- Smaller lot sizes
- Implementation of a pull system
- Better feedback
- Increased production
- Individual Responsibility
- Decreased WIP
- Dine flexibility

Before making changes, the team analyze each step in the original layout of the tube production line process.

1. There try to understand the original state process, identify the problem area, unnecessary step and non value added.

2. After mapping the process, the lean team collected data from the Material Review Board (MRB) bench to measure and analyze major types of defects . To better understand the process, the team also did a time study for 20 days period production run.

In the original state, the tube line consisted of one operator and four operations, separated into two stations by a large table using a push system. The table acted as a separator between the second and third operation.

The first problem discovered was the line's unbalanced . The first station was used about 70% of the time. Operators at the second station were spending a lot of their time waiting between cycle times. By combining stations one and two, room for improvement became evident with respect to individual responsibility, control of inventory by the operator, and immediate feedback when a problem occurred. The time study and the department layout reflect these findings.

A second problem was recognized. Because of the process flow, the production rate did not allow the production schedule to be met with two stations. Because operators lost track of machine cycles, machines were waiting for operator attention. Operators also tried to push parts through the first station—the bottleneck operation in the process—and then continued to manufacture the parts at the last two operations. Typically, long runs of WIP built up, and quality problems were not caught until a lot number of defective pieces were produced.

The original state data were taken from the last 20 days before the change. The teams analyze each step in the original and making changes. The findings of the time study on the original process provided the basis for reducing cycle time, balancing the line, designing the using Just In Time kanbans and scheduling, improve quality, decrease lot size and WIP , and improve flow. The new process data were taken starting one month after implementation. This delay gave the machine operators an opportunity to train and get to with the new process layout system.

With the U shaped cell design; The parts meet all the customer requirement. Table in the original process was removed ,almost eliminating WIP. With the reducing WIP and increasing production.

Some of the concepts used to improve the process included total employee involvement (TEI), smaller lot sizes, scheduling, point of use inventory, and improved layout. All employees and supervisors in the department were involved in all phases of the project. Their ideas and suggestions were incorporated in the planning and implementation process to gain wider acceptance of the changes to the process. Smaller lot sizes were introduced to minimize the number of parts produced before defects were detected. Kanbans were introduced (in the form of material handling racks) to control WIP and to implement a pull system. And the cell layout decreased travel between operations.

Operators were authorized to stop the line when problems arose. In the original-state , the operators were still continue running parts when a operation was down. With kanban

control, the layout eliminated the ability to store WIP, requiring the operator to shut down the entire line. The cell layout provides excellent opportunities for improving communication between operators about problems and adjustments, to achieve better quality.

Day-to-day inspection of the original-state process the operators spent a lot of time either waiting for material-handling person, or performing as a material handling. With the U-shaped cell, delivery to the point of use is more better for the operator. The operator places boxes of raw material on six moveable roller carts, where it's easily to get. The six boxes are enough to last a 24-hr period.

To reduce setup times, tools needed for machine repair and adjustments are located in the cell. The screws are not standardized; tools are set up in order of increasing size to quickly identify the proper tool.

For three months the process was monitored to verify that it was in control. Comparison of time studies from the original-state and the implemented layout demonstrated an increase in production from 300 to 514 finished products per shift. The new layout eliminated double handling between the second and third operations, as well as at the packing step. It also reduced throughout time by making it easier to cycle all four operations in a pull-system order. Customer demand was met by two shifts, which reduced the labor cost.

The results of the redesign are as follows:

- WIP decreased by 97%
- Production increased 72%
- Scrap was reduced by 43%
- Machine utilization increased by 50%
- Labor utilization increased by 25%
- Labor costs were reduced by 33%
- Sigma level increased from 2.6 to 2.8

This project yielded reduced labor and scrap costs, and allowed the organization to do a better job of making deliveries on time, while allowing a smaller finished-goods inventory. Daily production numbers and single-part cycle time served as a benchmark for monitoring progress towards the goal. Although the sigma level increase , the 43% reduction in defects, 97% reduction in WIP, and production increase of 72% contributed to the project objective.

Implementing lean is a never ending process; this is what continuous improvement is all

about. When you get one aspect of lean implemented, it can always be improved. Don't get hung up on it, but don't let things slip back to the starting point. There will always be time to go back and refine some of the processes.

Before Lean Manufacturing was implemented at Nypro Oregon Inc., we would operate using traditional manufacturing. Traditional manufacturing consists of producing all of a given product for the marketplace so as to never let the equipment idle. These goods them need to be warehoused or shipped out to a customer who may not be ready for them. If more is produced than can be sold, the products will be sold at a deep discount (often a loss) or simply scrapped. This can add up to an enormous amount waste. After implementing Lean Manufacturing concepts, our company uses just in time. Just in time refers to producing and delivering good in the amount required when the customer requires it and not before. In lean Manufacturing, the manufacture only produces what the customer wants, when they want it. This often a much more cost effective way of manufacturing when compared to high priced, high volume equipment.

Unit Test:

1. What is 5S?
2. Please Explain each "S" of the 5S.
3. Please Explain Kaizen concept.
4. What is the Pull processing?
5. What is the Poka-yoke?

6. What is the six-sigma DMAIC?
7. What is the objectives for a new process layout of the tube production line?
8. Before making changes, The Manufacturing engineers team do what first?
9. Please lists the results of the redesign.
10. The key to implementing lean new idea or concept is to do what?

CHAPTER ATTRIBUTION INFORMATION

This chapter was derived from the following sources.

PART VIII

Chapter 8: CNC

18

Chapter 8: CNC

Unit 1: Introduction to CNC

What is CNC? CNC is a Computer Numerical Control. CNC is the automation of machine tools that are operated by precisely programmed commands encoded and played by a computer as opposed to controlled manually via handwheels or levers.

In modern CNC systems, end-to-end component design is highly automated using Computer-Aided Design (CAD) and Computer-Aided Manufacturing (CAM) programs. The series of steps needed to produce any part is highly automated and produces a part that closely matches the original CAD design.

In the CNC machines the role of the operators is minimized. The operator has to merely feed the program of instructions in the computer, load the required tools in the machine, and rest of the work is done by the computer automatically. The computer directs the machine tool to perform various machining operations as per the program of instructions fed by the operator.

The CNC technology can be applied to wide variety of operations like drafting, assembly, inspection, sheet metal working, etc. But it is more prominently used for various metal machining processes like turning, drilling, milling, shaping, etc. Due to the CNC, all the machining operations can be performed at the fast rate resulting in bulk manufacturing becoming quite cheaper.

How It Works

The CNC machine comprises of the computer in which the program is fed for cutting of the metal of the job as per the requirements. All the cutting processes that are to be carried out and all the final dimensions are fed into the computer via the program. The computer thus knows what exactly is to be done and carries out all the cutting processes. CNC machine works like the Robot, which has to be fed with the program and it follows all your instructions.

You don't have to worry about the accuracy of the job; all the CNC machines are designed to meet very close accuracies. In fact, these days for most of the precision jobs CNC machine is compulsory. When your job is finished, you don't even have to remove it, the machine does that for you and it picks up the next job on its own. This way your machine can keep on doing the fabrication works all the 24 hours of the day without the need of much monitoring, of course you will have to feed it with the program initially and supply the required raw material.

Since the earliest days of production manufacturing, ways have been sought to increase dimensional accuracy as well as speed of production. Simply put, numerical control is a method of automatically operating a manufacturing machine based on a code of letter, numbers, and special characters. As they developed, application of digital computers control of manufacturing equipment was realized. Computers were soon used to provide direct control of machine tools. The integrated circuit led to small computers used to control individual machines, and the computer numerical control (CNC) era was born. This Computer Numerical Control era has become so sophisticated it is the preferred method of almost every phase of

precision manufacturing, particularly machining. Precision dimensional requirements, mainstay of the machining processes, are ideal candidates for use of computer control systems. Computer numerical control now appears in many other types of manufacturing processes. A distinct advantage of computer control of machine tools is rapid, high-precision positioning of workpiece and cutting tools.

Today, manual machine tools have been largely replaced by Computer numerical Control(CNC) machine tools. The machine tool are controlled electronically rather than by hand. CNC machine tools can produce the same part over and over again with very little variation. Modern CNC machines can position cutting tools and workpieces at traverse feed rates of several hundred inches per minute, to an accuracy of .0001". Once programming is complete and tooling is set up, they can run day or night, week after week, without getting tried, with only routine service and cutting tool maintenance. These are obvious advantage over manual machine tools, which need a great deal of human interaction in order to do anything. Cutting feed rates and spindle speeds may be optimized through program instructions. Modern CNC machine tools have turret or belt toolholders and some can hold more than 150 tools. Tool change take less than 15 seconds.

Computer Numerical Control machine are highly productive. They are also expensive to purchase, set up, and maintain. However, the productivity advantage can easily offset this cost if their use is properly managed. A most important advantage of CNC is ability to program the machine to do different jobs. Tool selection and changing under program control is extremely productive, with little time wasted applying a tool to the job.

A program developed to accomplish a given task may be used for a short production run of one, or a few parts. The machine may then be set up for a new job and used for long production runs of hundreds or thousands of production units. It can be interrupted, used for the original job or another new job, and quickly returned to the long production run. This makes the CNC machine tool extremely versatile and productive. Computer-aided design(CAD), has become the preferred method of product design & development. The connection between CAD & CNC was logical. A computer part design can go directly to program used to develop CNC machine control information. A CNC manufacturing machine can then make the part. The computer is extremely useful for assisting the CNC programmer in developing a program to manufacture a specific part. Computer-aided manufacturing, or CAM, systems are now the industry standard for programming. When CAD, CAM & CNC are blended, the greatest capability emerges, producing parts extremely difficult or impossible to make by manual methods.

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

To fully understand numerical control programming you must understand axes and coordinates. Think of a part that you would have make. You could describe it to someone else by its geometry. For example, the part you have make is a 5 inch by 8 inch rectangle. All parts can be described in this fashion. Any point on the machined part, such as a pocket to be cut or a hole to be drilled, can be described in term of its position. The system that allows us to do this, called the Cartesian Coordinate or rectangular coordinate system.

19

Unit 2: CNC Machine Tool Programmable Axes and Position Dimensioning Systems

OBJECTIVE

After completing this unit, you should be able to:

- Understand the cartesian coordinate system.
- Understand the Cartesian coordinates of the plane.
- Understand the Cartesian coordinates of three-dimensional space.
- Understand the four Quadrants.
- Explain the difference between polar and rectangular coordinated.
- Identify the programmable axes on a CNC machining.

THE CARTESIAN COORDINATE SYSTEM

Cartesian coordinates allow one to specify the location of a point in the plane, or in three-dimensional space. The Cartesian coordinates or rectangular coordinates system of a point are a pair of numbers (in two-dimensions) or a triplet of numbers (in three-dimensions) that specified signed distances from the coordinate axis. First we must understand a coordinate system to define our directions and relative position. A system used to define points in space by establishing directions(axis) and a reference position(origin). A coordinate system can be rectangular or polar.

Just as points on the line can be placed in one to one correspondence with the real number line, so points in plane can be placed in one to one correspondence with pairs of real number line by using two coordinate lines. To do this, we construct two perpendicular coordinate line that intersect at their origins; for convenience. Assign a set of equally space graduations to the x and y axes starting at the origin and going in both directions, left and right (x axis) and up and down (y axis) point along each axis may be established. We make one of the number lines vertical with its positive direction upward and negative direction downward. The other number lines horizontal with its positive direction to the right and negative direction to the left. The two number lines are called coordinate axes; the horizontal line is the x axis, the vertical line is the y axis, and the coordinate axes together form the Cartesian coordinate system or a rectangular coordinate system. The point of intersection of the coordinate axes is denoted by O and is the origin of the coordinate system. See Figure 1.

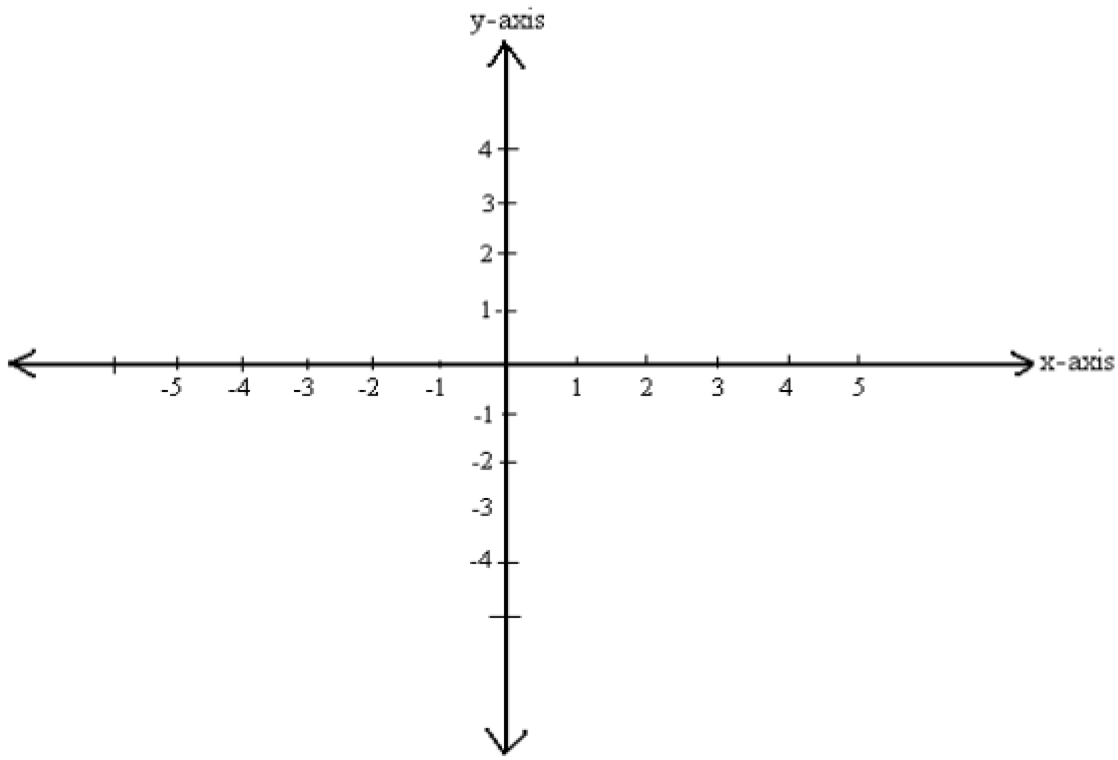


Figure 1

It is basically, Two Real Number Lines Put Together, one going left-right, and the other going up-down. The horizontal line is called x-axis and the vertical line is called y-axis.

The Origin

The point (0,0) is given the special name "The Origin", and is sometimes given the letter "O".

Real Number Line

The basis of this system is the real number line marked at equal intervals. The axis is labeled (X, Y or Z). One point on the line is designated as the Origin. Numbers on one side of the line are marked as positive and those to the other side marked negative. See Figure 2.

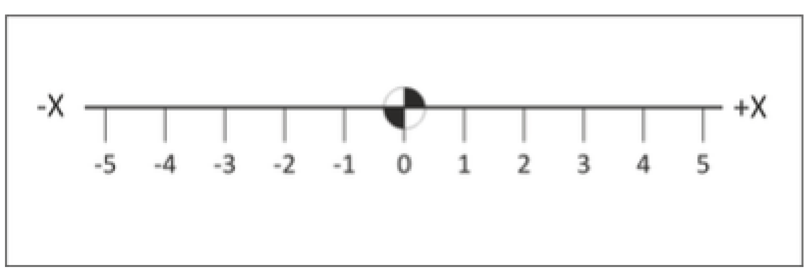


Figure 2. X-axis number line

Cartesian coordinates of the plane

A plane in which a rectangular coordinate system has been introduced is a coordinate plane or an x-y-plane. We will now show how to establish a one to one correspondence between points in a coordinate plane and pairs of real number. If A is a point in a coordinate plane, then we draw two lines through A, one perpendicular to the x-axis and one perpendicular to the y-axis. If the first line intersects the x-axis at the point with coordinate x and the second line intersects the y-axis at the point with coordinate y, then we associate the pair (x,y) with the A(See Figure 2). The number a is the x-coordinate or abscissa of P and the number b is the y-coordinate or ordinate of p; we say that A is the point with coordinate (x,y) and denote the point by A(x,y). The point (0,0) is given the special name "The Origin", and is sometimes given the letter "O".

Abscissa and Ordinate:

The words "Abscissa" and "Ordinate" … they are just the x and y values:

- Abscissa: the horizontal ("x") value in a pair of coordinates: how far along the point is.
- Ordinate: the vertical ("y") value in a pair of coordinates: how far up or down the point is.

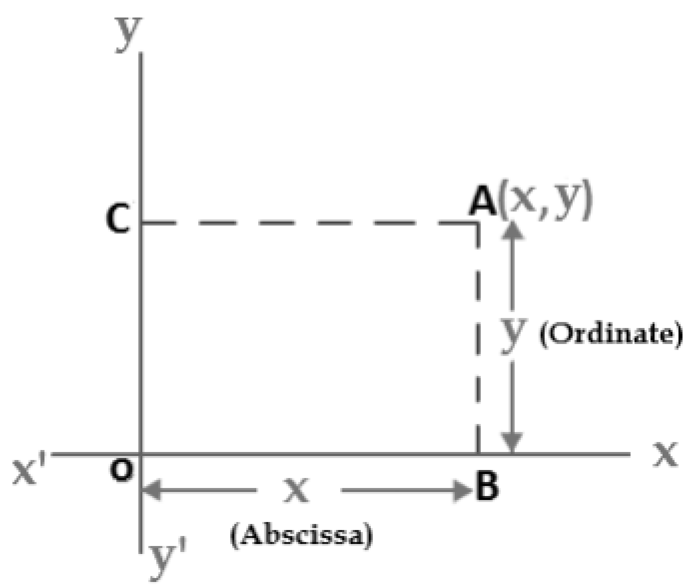


Figure 3

Negative Values of X and Y:

The Real Number Line, you can also have negative values.

Negative: start at zero and head in the opposite direction; See Figure 4

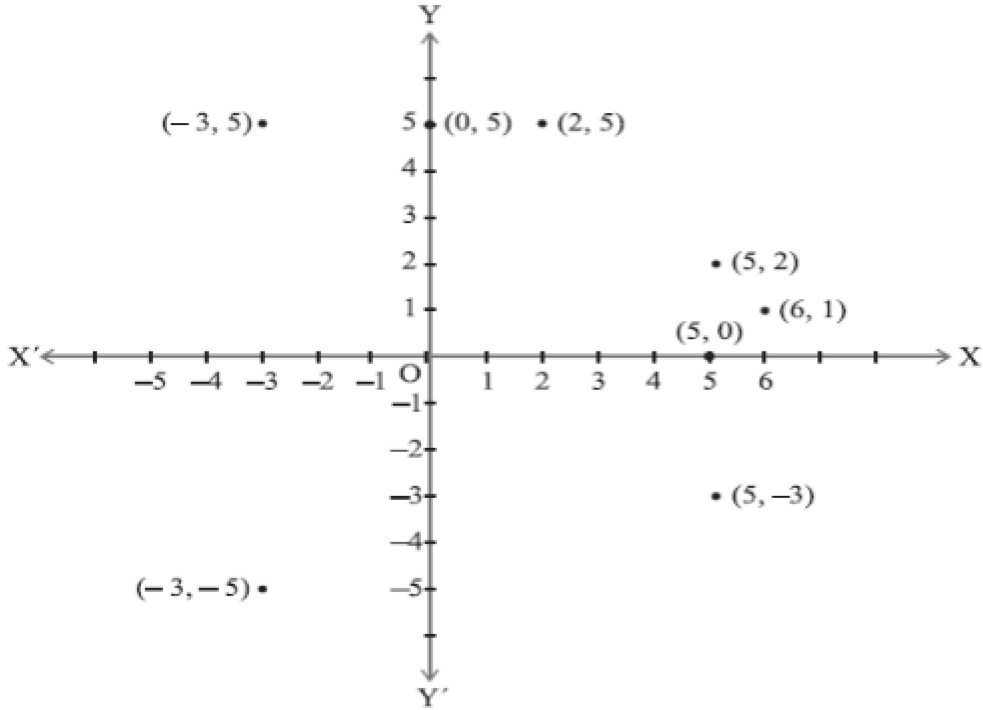


Figure 4

So, for a negative number:

- go left for x
- go down for y

For example (-3,5) means:
go left along the x axis 3 then go up 5 in the y-axis. (Quadrant II x is negative ,y is positive)
And (-3,-5) means:
go left along the x axis 3 then go down 5 in the y-axis. (Quadrant III x is negative ,y is negative)
Using Cartesian Coordinates, mark a point on a graph by how far along and how far up it is; See figure 5. The point (12,5) is 12 units along the x-axis, and 5 units up on the y-axis.

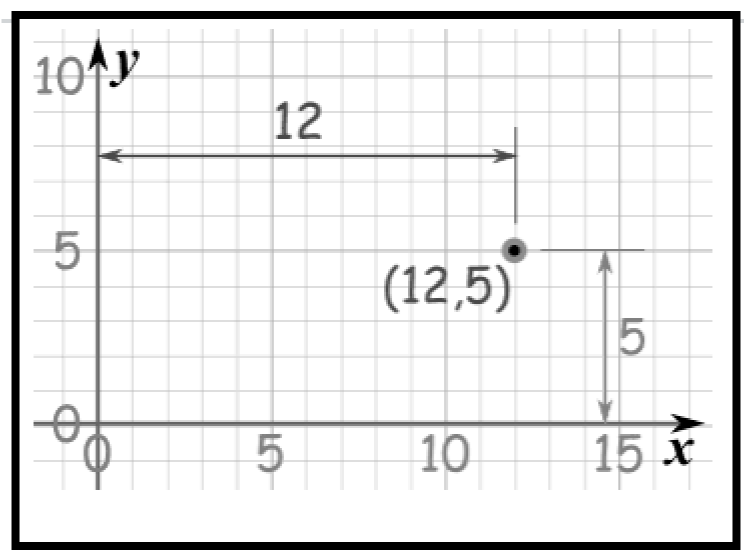
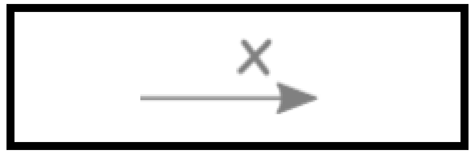
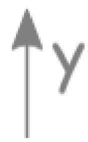


Figure 5

X and Y Axis:



The horizontal line is called x-axis and the vertical line is called y-axis; both line runs throughzero(Origin, (0,0)).



The horizontal line is called x-axis and the vertical line is called y-axis; both line runs throughzero(Origin, (0,0)).
Put them together on a graph …See figure 6

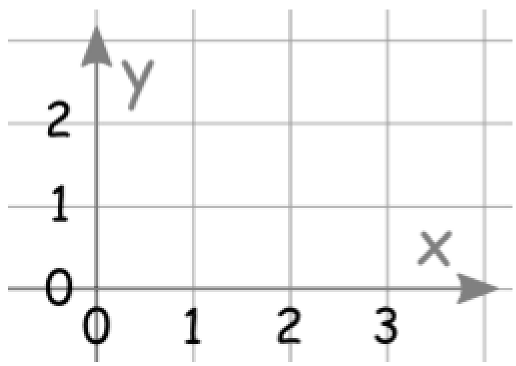
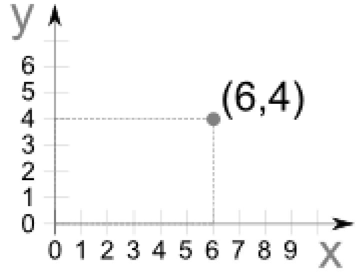


Figure 6

It is basically, a set of two Real Number lines.
Axis: The reference line from which distances are measured.

Example:



Point (6,4) is

Go along the x direction 6 units then go up 4 units up in the y direction then "plot the dot".

And you can remember which axis is which by:

The coordinates are always written in a certain order:

- the horizontal distance first,
- then the vertical distance.

Ordered pair:

The numbers are separated by a comma, and parentheses are put around the whole thing like this: (7,4)

Example: (7,4) means 7 units to the right(x-axis), and 4 units up(y-axis)

Cartesian coordinates of three-dimensional space

In three-dimensional space(xyz space), oriented at right angles to the xy-plane. The z axis, passes through the origin of the xy-plane. Coordinates are determined according to the east-west for x-axis north-south for y-axis, and up-down for the z-axis displacements from the origin. The Cartesian coordinate system is based on three mutually perpendicular coordinate axes: the x-axis, the y-axis, and the z-axis, See Figure 6 below. The three axes intersect at the point called the origin. You can imagine the origin being the point where the walls in the corner of a room meet the floor. The x-axis is the horizontal line along which the wall to your left and the floor intersect. The y-axis is the horizontal line along which the wall to your right and the floor intersect. The z-axis is the vertical line along which the walls intersect. The parts of the lines that you see while standing in the room are the positive portion of each of the axes. The negative part of these axes would be the continuations of the lines outside of the room.

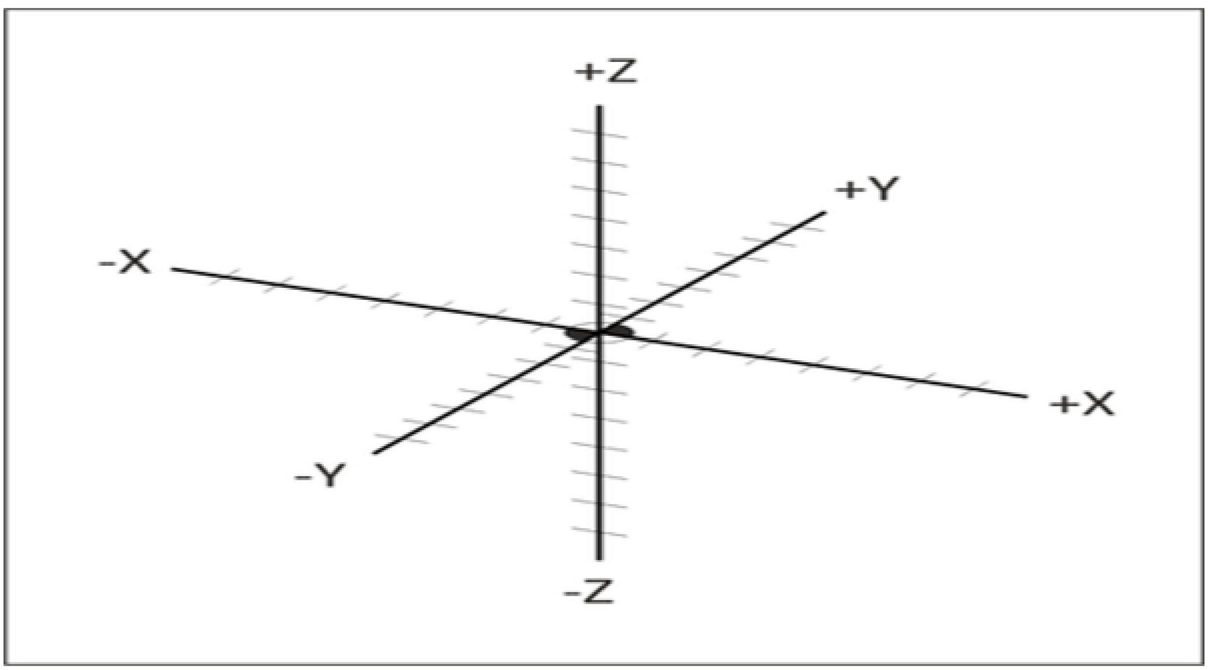


Figure 7. 3D Cartesian Coordinate System

Three-dimensional Cartesian coordinate axes. A representation of the three axes of the three-dimensional Cartesian coordinate system. The positive x-axis, positive y-axis, and positive z-axis are the sides labeled by x, y and z. The origin is the intersection of all the axes. The branch of each axis on the opposite side of the origin (the unlabeled side) is the negative part.

When dealing with 3-dimensional motion, is to set up a suitable coordinate system. The most straight-forward type of coordinate system is called a Cartesian system. A Cartesian coordinate system consists of three mutually perpendicular axes, the X, Y, and Z-axes. By convention, the orientation of these axes is such that when the index finger, the middle finger, and the thumb of the right-hand are configured so as to be mutually perpendicular, the index finger, the middle finger, and the thumb can be aligned along the X, Y, and Z-axes, respectively. Such a coordinate system is termed right-handed. See Figure 7. The point of intersection of the three coordinate axes is termed the origin of the coordinate system.

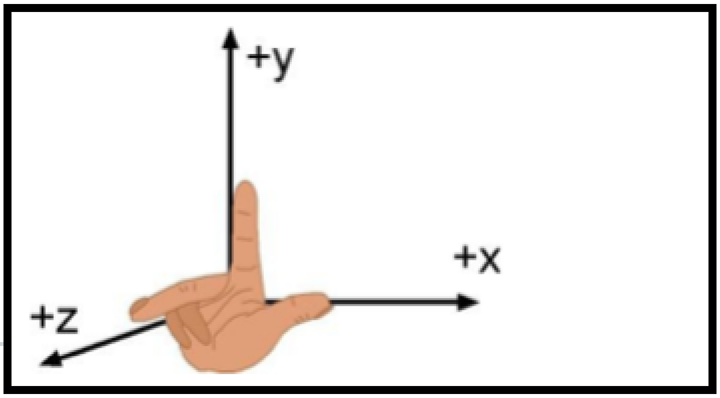


Figure 8. The Right Handed Cartesian System

The Cartesian coordinates of a point in three dimensions are a triplet of numbers (x,y,z). The three numbers, or coordinates, specify the signed distance from the origin along the x, y, and z-axes, respectively. They can be visualized by forming the box with edges parallel to the coordinate axis and opposite corners at the origin and the given point.

The points may now be defined in a three dimensional volume of space. This permits to define points in three dimensions from the origin. The Cartesian coordinates (x,y,z) of a point in three-dimensions specify the signed distance from the origin along the x, y, and z-axes, respectively. Z-axis points become the third entry when defining coordinate locations.

Given the above corner-of-room analogy, we could form the Cartesian coordinates of the point at the top of your head, as follows. Imagine that you are five meters tall the z-axis, and that you walk two meters from the origin along the x-axis, then turn left and walk parallel to the y-axis four meters into the room. The Cartesian coordinates of the point at the top of your head would be (2,4,5).

For example, a notation of (2,4,5) corresponds to the value of X2, Y4, and Z5. See Figure 8.

3 Dimensions

Cartesian coordinates can be used for locating points in 3 dimensions as in this example:

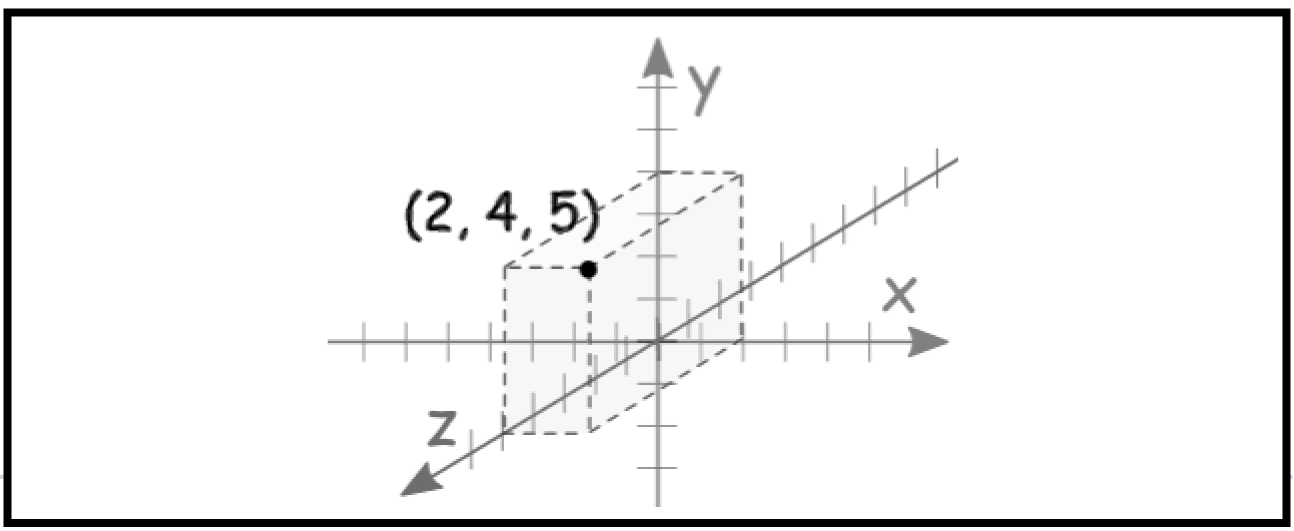


Figure 9. The point (2, 4, 5) is shown in three-dimensional Cartesian coordinates.

Quadrants

The coordinate axes divide the plane into four parts, called quadrants (See Figure 9). The quadrants are number counterclockwise, starting from the upper right, labeled I, II, III and IV with axes designations as shown in the illustration below.

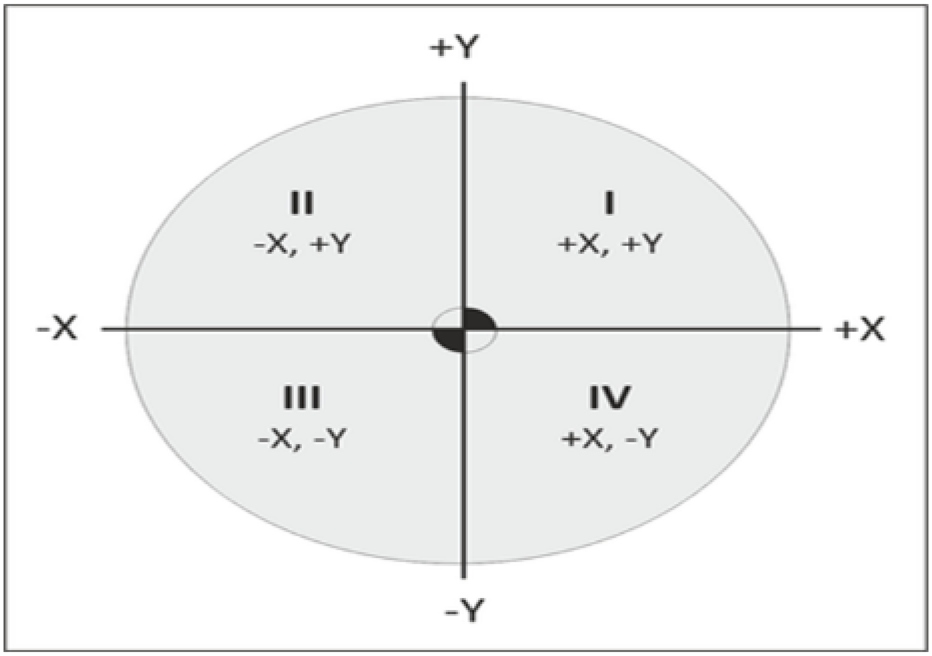


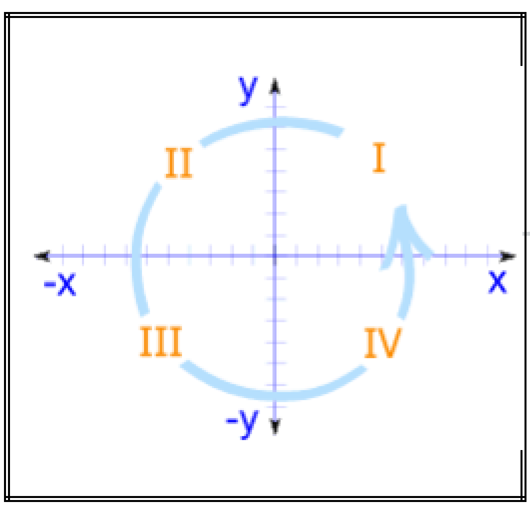
Figure 10

Four Quadrants:

When we include negative values, the x and y axes divide the space up into 4 pieces:

Quadrants I, II, III and IV

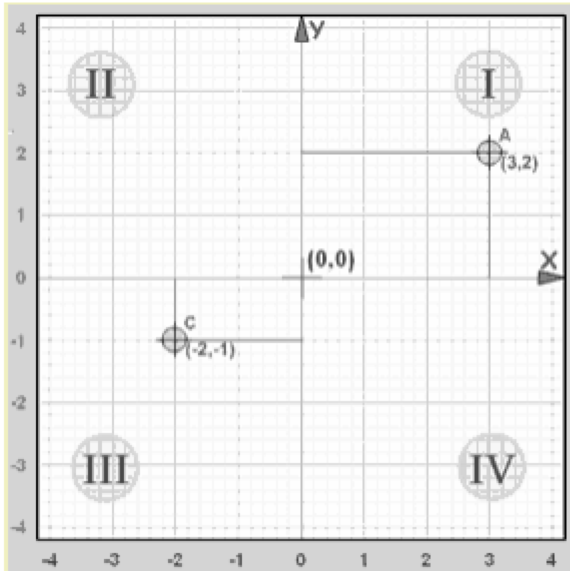
(They are numbered in a counterclockwise direction)



In Quadrant I : both x and y are positive

In Quadrant II : x is negative (y is still positive)

In Quadrant III : both x and y are negative
In Quadrant IV : x is positive again, while y is negative



Quadrant	X (Horizontal)	Y (Vertical)
I	Positive	Positive
II	Negative	Positive
III	Negative	Negative
IV	Positive	Negative

Example: The point "A" (3,2) is 3 units along the x-axis, and 2 units up the y-axis.

Both x and y are positive, so that point is in "Quadrant I"

Example: The point "C" (-2,-1) is 2 units along the x-axis in the negative direction, and 1 unit down the y-axis in the negative direction.

Both x and y are negative, so that point is in "Quadrant III"

Dimensions: 1, 2, 3 and more …

1. The Real Number Line can only go:
   - left-right
   - so any position needs just one number
2. Cartesian coordinates can go:
   - left-right, and
   - up-down
   - so any position needs two numbers
3. 3 dimensions

- left-right,
- up-down, and
- forward-backward

UNIT TEST

1. What is CNC?

2. Describe the cartesian coordinate system.

3. What is The Origin?

4. The Horizontal line is called what?

5. The Vertical line is called what?

6. Describe the real number line.

7. Explain Abscissa and Ordinate.

8. What are the representation of the three axes of the three dimensional cartesian coordinate system.

9. The coordinate axes divide the plane into four parts, is called what?

10. In Quadrant IV, the X axes and the Y axes is what?

20

Unit 3: Vertical Milling Center Machine Motion

OBJECTIVE

After completing this unit, you should be able to:

- Understand the Vertical Milling Center Machine Motion.
- Understand the Machine Home Position.
- Understand the CNC Machine Coordinates.
- Understand the Work Coordinate System.
- Understand the Machine and Tool Offsets.
- Set Tool Length Offset for each tool.

VMC Machine Motion

CNC machines use a 3D Cartesian coordinate system. Figure 10. shows a typical Vertical Milling Center (VMC). Parts to be machined is fastened to the machine table. This table moves in the XY-Plane. As the operator faces the machine, the X-Axis moves the table left-right. The Y-Axis moves the table forward-backward. The machine column grips and spins the tool. The column controls the Z-axis and moves up-down.

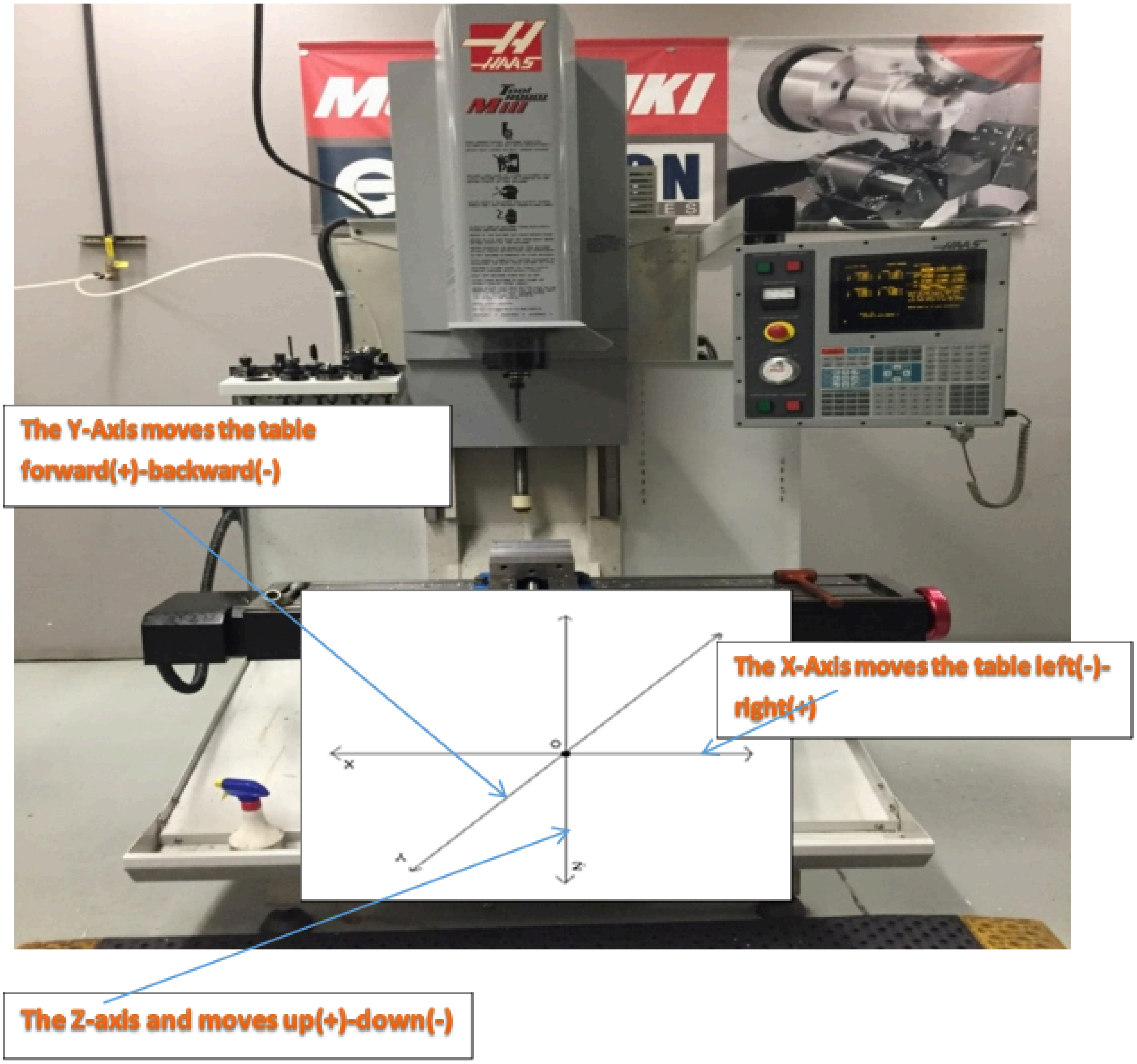


Figure 1.VMC Machine Motion

CNC Machine Coordinates

The CNC Machine Coordinate System is illustrated in Figure 11. The control point for the Machine Coordinate System is defined as the center-face of the machine spindle. The Origin point for the machine coordinate system is called Machine Home. This is the postion of the center-face of the machine spindle when the Z-axis is fully retracted and the table is moved to its limits near the back-left corner.

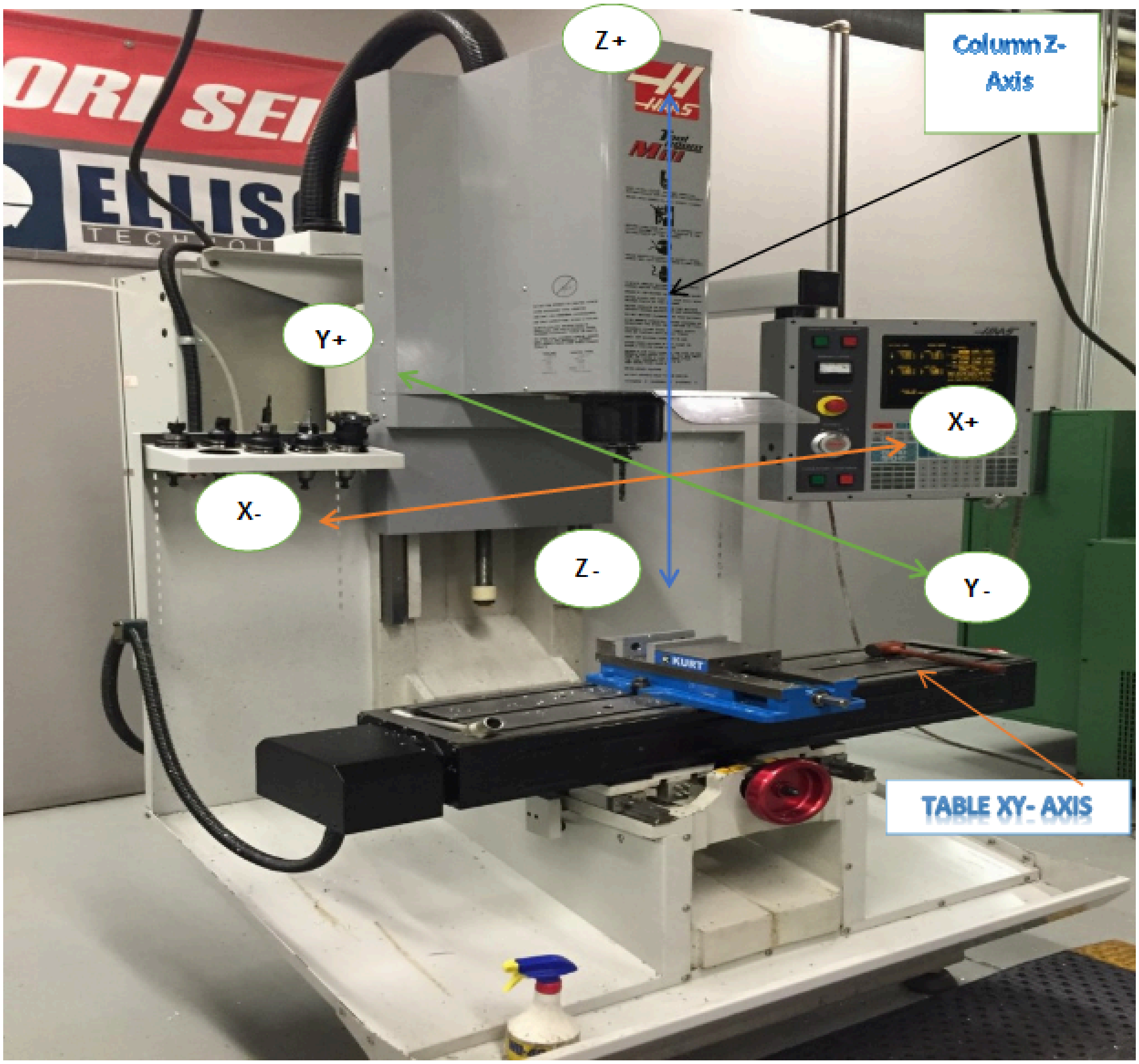


Figure 2. VMC Machine Coordinate System (At Home Position)

As shown in Figure 12, when working with a CNC, always think, work, and write CNC programs in terms of tool motion, not table motion. For example, increasing +X coordinate values move the tool right in relation to the table (though the table actually moves left). Likewise, increasing +Y coordinate values move the tool towards the back of the machine (the table moves towards the operator). Increasing +Z commands move the tool up (away from the table).

About Machine Home Position

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

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

Work Coordinate System

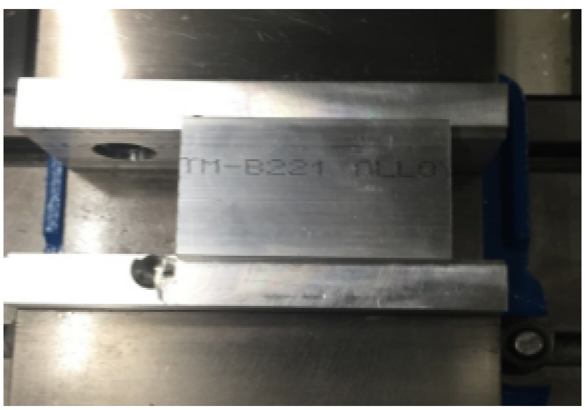
Obviously it would be difficult to write a CNC program in relation to Machine Coordinates. The home position is far away from the table, so values in the CNC program would be large and have no easily recognized relation to the part model. To make programming and setting up the CNC easier, a Work Coordinate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

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

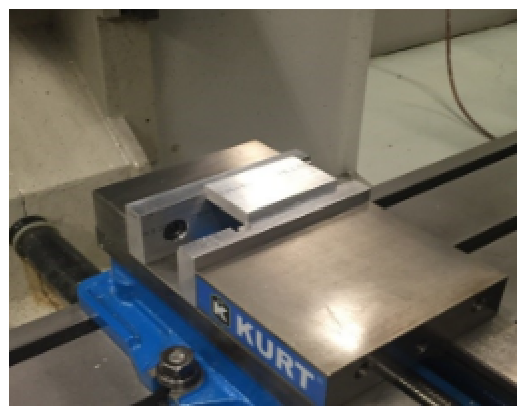
- The WCS location must be able to be found by mechanical means such as an edge finder, coaxial indicator or part probe.
- It must be located with high precision: typically plus or minus .001 inches or less.
- It must be repeatable: parts must be placed in exactly the same position every time.
- It should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

For example, Figure 13 shows a part gripped in a vise. The outside dimensions of the part have already been milled to size on a manual machine before being set on the CNC machine.

The CNC is used to make the holes, pockets, and slot in this part. The WCS is located in the upper-left corner of the block. This corner is easily found using an Edge Finder or Probe.



Top View



Side View

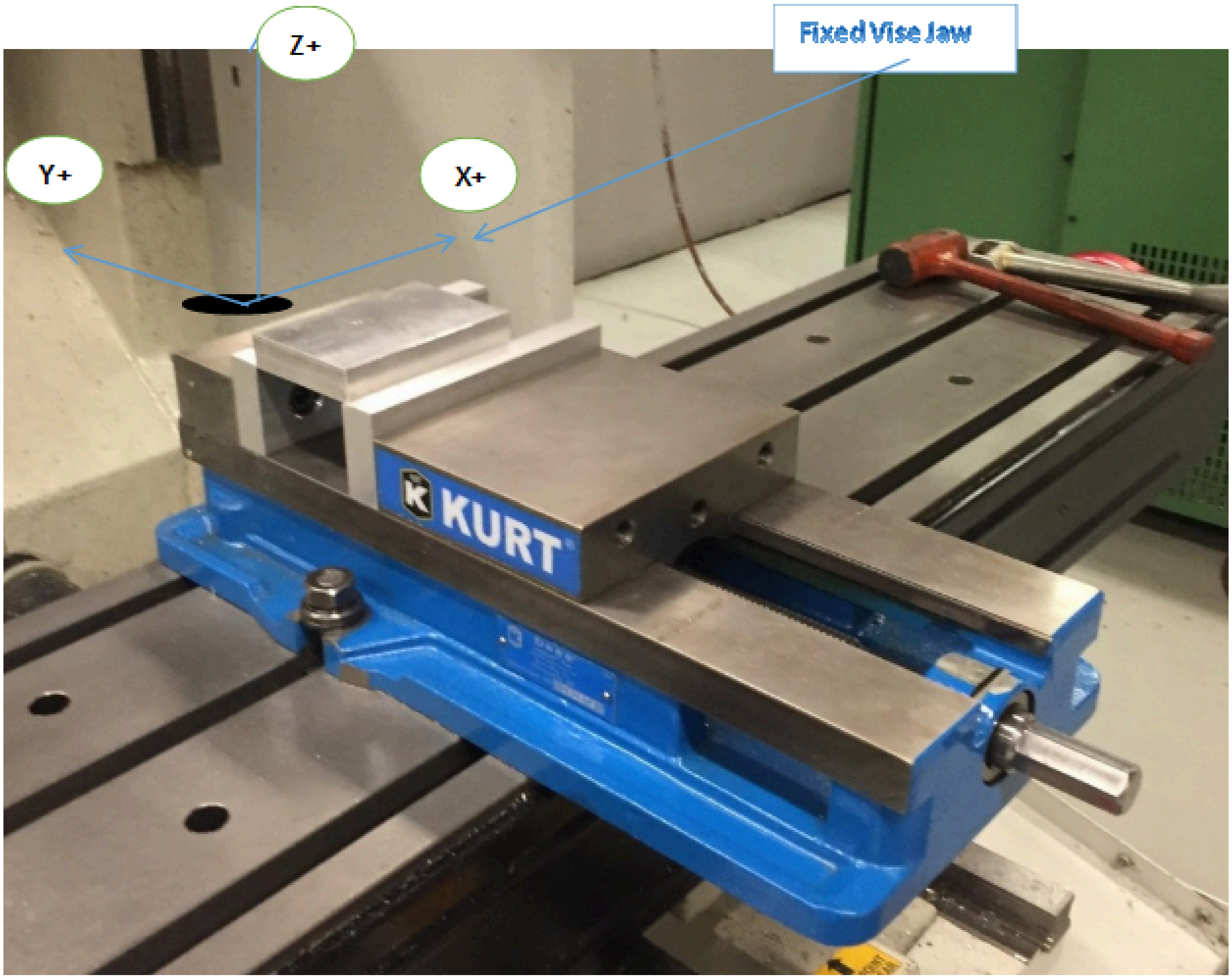


Figure 3. Work Coordinate System (WCS)

Machine and Tool Offsets

Machine Offsets:

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

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

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

Part Offset XY:

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

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

Notice in Figure 14 how Part Offsets (+X, -Y) are used to shift the centerline of the machine spindle directly over the WCS.

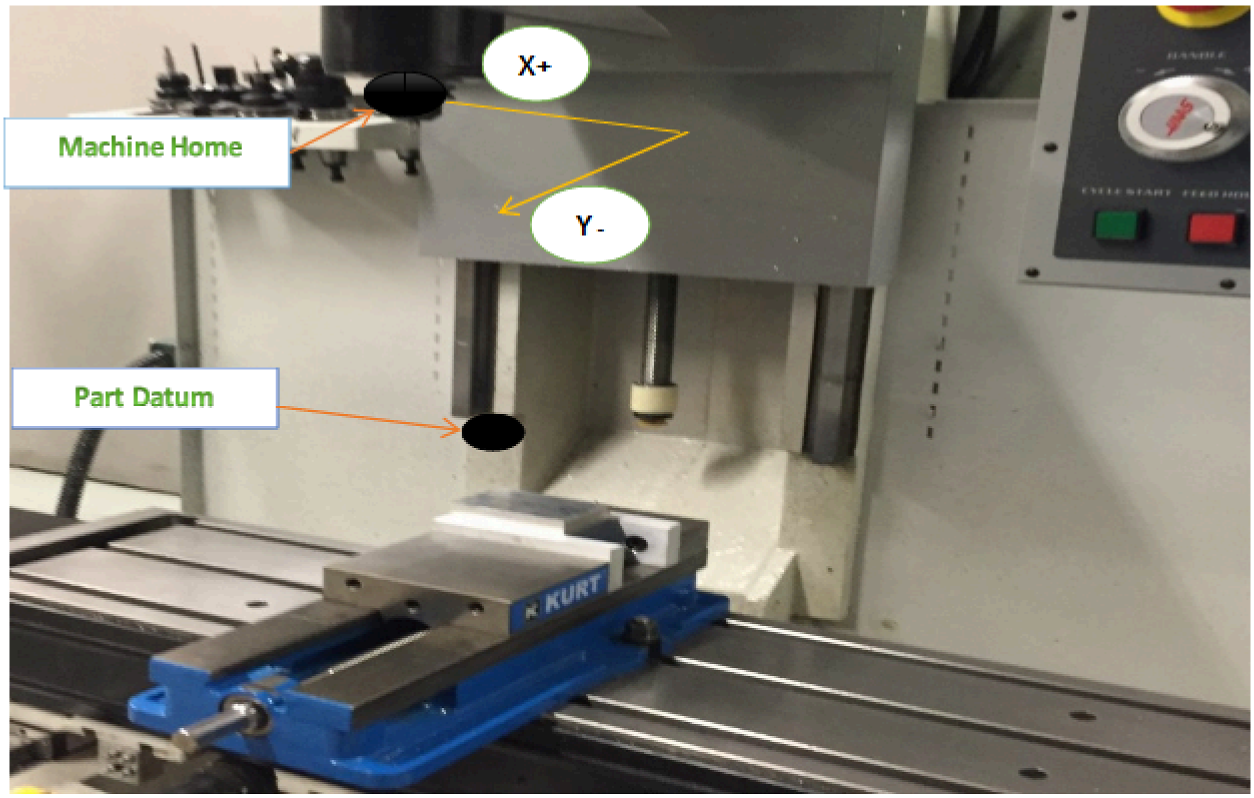


Figure 4: PartOffset Shifts Machine to WCS

Part Offset Z

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

Tool Length Offset (TLO)

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

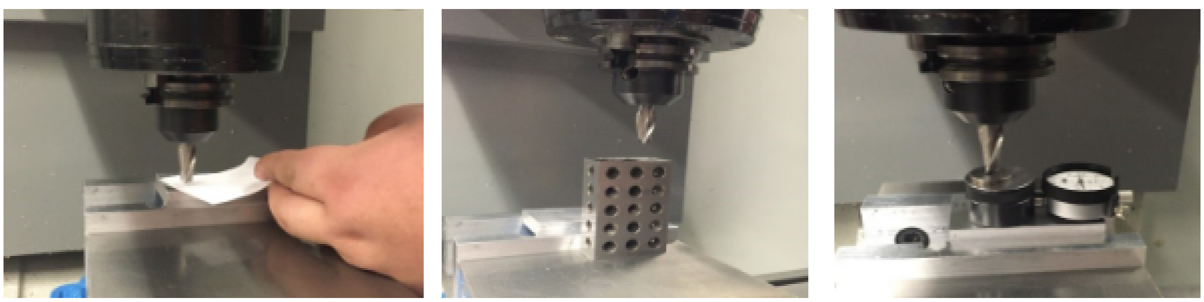
In its simplest use, the TLO is found by jogging the spindle with tool from the machine home Z-position to the part Z-zero position, as shown on the far left in Figure 17 below. The tool is jogged to the part datum Z and the distance travelled is measured. This value is entered in the TLO register for that tool. Problems with this method include the need to face mill the part to the correct depth before setting tools. Also, if the Z-datum is cut away (typical of 3D surfaced parts) it is impossible to set the datum should a tool break or wear and need to be replaced. All tools must be reset whenever a new job is set up. When this method is used, the Fixture Offset Z is not used, but set to zero.

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

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

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

3-Ways to Set Tool Length Offset



Part Datum(Easiest) 1-2-3 Block(Better) Tool Probe(Best)

Figure 5. Ways to set TLO

UNIT TEST

1. Explain Machine home position.

2. On the Vertical Milling Center(VMC) the X axis move the table in what direction.

3. On the Vertical Milling Center(VMC) the Y axis move the table in what direction.

4. On the Vertical Milling Center(VMC) the Z axis move the table in what direction.

5. Please lists 3-ways to set Tool length offsets.

21

Unit 4: CNC Language and Structure

OBJECTIVE

After completing this unit, you should be able to:

- Identify the programs list instructions.
- Understand the Program Format
- Describe Letter Address Commands codes
- Describe Special Character Code Definitions.
- Understand the G & M Codes.

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

The blocks are arranged in the following order:

- Program Start
- Load Tool
- Spindle On
- Coolant On
- Rapid to position above part
- Machining operation

- Coolant Off
- Spindle Off
- Move to safe position
- End program

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

Table 3 and Table 4 in section **G & M Codes** show the most common G and M codes that should be memorized if possible.

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

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

Program Format

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

Block	Description	Purpose
%	Start of program.	Start Program
O1234	Program number (Program Name).	
(T1 0.25 END MILL)	Tool description for operator.	
G17 G20 G40 G49 G80 G90	Safety block to ensure machine is in safe mode.	
T1 M6	Load Tool #1.	Change Tool
S9200 M3	Spindle Speed 9200 RPM, On CW.	
G54	Use Fixture Offset #1.	Move to Position
M8	Coolant On.	
G00 X-0.025 Y-0.275	Rapid above part.	
G43 Z1.H1	Rapid to safe plane, use Tool Length Offset #1.	
Z0.1	Rapid to feed plane.	
G01 Z-0.1 F18.	Line move to cutting depth at 18 IPM.	
G41 Y0.1 D1 F36.	CDC Left, Lead in line, Dia. Offset #1, 36 IPM.	Machine Contour
Y2.025	Line move.	
X2.025	Line move.	
Y-0.025	Line move.	
X-0.025	Line move.	
G40 X-0.4	Turn CDC off with lead-out move.	
G00 Z1.	Rapid to safe plane.	
M5	Spindle Off.	Change Tool
M9	Coolant Off.	
(T2 0.25 DRILL)	Tool description for operator.	
T2 M6	Load Tool #2.	
S3820 M3	Spindle Speed 3820 RPM, On CW.	

M8	Coolant On.	
X1. Y1.	Rapid above hole.	Move to Position
G43 Z1.H2	Rapid to safe plane, use Tool Length Offset 2.	
Z0.25	Rapid to feed plane.	
G98 G81 Z-0.325 R0.1 F12.	Drill hole (canned) cycle, Depth Z-.325, F12.	
G80	Cancel drill cycle.	Drill Hole
Z1.	Rapid to safe plane.	
M5	Spindle Off.	
M9	Coolant Off.	
G91 G28 Z0	Return to machine Home position in Z.	
G91 G28 X0 Y0	Return to machine Home position in XY.	End Program
G90	Reset to absolute positioning mode (for safety).	
M30	Reset program to beginning.	
%	End Program.	

Letter Address Commands codes

The command block controls the machine tool through the use of letter address commands. Some are used more than once, and their meaning changes based on which G-code appears in the same block.

Codes are either modal, which means they remain in effect until cancelled or changed, or non-modal, which means they are effective only in the current block. As you can see, many of the letter addresses are chosen in a logical manner (T for tool, S for spindle, F for feed rate, etc.).

The table below lists the most common Letter Address Commands codes.

Table 2: Letter Address Commands Codes

Variable	Description	Definitions
A	Absolute or incremental position ofAaxis (rotational axis around X axis)	A,B,C – 4th/5th Axis Rotary Motion Rotation about the X, Y or Z-axis respectively.The angle is in degrees and up to three decimal places precision. G01 A45.325B90.
B	Absolute or incremental position of B axis (rotational axis around Y axis)	Same as A
C	Absolute or incremental position of C axis (rotational axis around Z axis)	Same as B
D	Defines diameter or radial offset used for cutter compensation	Used to compensate for tool diameter wear and deflection.D is accompanied by an integer that isthe same as the tool number (T5 uses D5,etc). No decimal point is used. Itis always usedin conjunction with G41 or G42 and a XY move (never an arc). When called, the control reads the register and offsets the tool path left (G41) or right (G42) by the value in the register. G01 G41 X2.D1
E	Precision feed rate for threading on lathes	
F	Defines feed rate	Sets the feed rate when machining lines, arcs or drill cycles.Feed rate can be in Inches per Minute (G94 mode) or Inverse Time (G93 mode). Feed rates can be up to three decimal placesaccuracy (for tap cycles) and require a decimal point. G01 X2.Y0. F30.
G	Address for preparatory commands	G commands often tell the control what kind of motion is wanted (e.g., rapid positioning, linear feed, circular feed, fixed cycle) or what offset value to use. G02 X2.Y2.I.50J0.
H	Defines tool length offset; Incremental axis corresponding to C axis (e.g., on a turn-mill)	This code calls a tool length offset (TLO) register on the control. The control combines the TLO and Fixture Offset Z values to know where the tool is in relation to the part datum.It is always accompanied by an integer (H1, H2,etc), G43, and Z coordinate. G43 H1 Z2.

I	Defines arc size inX axisfor G02 or G03 arc commands. Also used as a parameter within some fixed cycles.	For arc moves (G2/G3), this is the incremental X-distance from the arc start point to the arc center. Certain drill cycles also use I as an optional parameter. G02 X.5 Y2.500I0.J0.250
J	Defines arc size inY axisfor G02 or G03 arc commands. Also used as a parameter within some fixed cycles.	For arc moves (G2/G3), this is the incremental Y-distance from the arc start point to the arc center. Certain drill cycles also use J as an optional parameter. G02 X.5 Y2.500 I0.J0.250
K	Defines arc size inZ axisfor G02 or G03 arc commands. Also used as a parameter within some fixed cycles, equal to L address.	For anarcmove (G2/G3) this is the incremental Z-distance from the arc start point to the arc center. In the G17 plane, this is the incremental Z-distance for helical moves. Certain drill cycles also use J as an optional parameter. G18 G03 X.3 Z2.500 I0.K0.250
L	Fixed cycle loop count; Specification of what register to edit using G10	Fixed cycle loop count: Defines number of repetitions ("loops") of a fixed cycle at each position.Assumed to be 1 unless programmed with another integer.Sometimes the K addressis usedinstead of L. With incremental positioning (G91), a series of equally spaced holes can be programmed as a loop rather than as individual positions.G10 use:Specificationof what register to edit (work offsets, tool radius offsets, tool length offsets, etc.).
M	Miscellaneous function	Always accompanied by an integer that determines its meaning.Only one M-codeis allowedin each block of code. Expanded definitions of M-codes appear later in this chapter. M08
N	Line (block) number in program; System parameter number to be changed using G10	Block numbers can make the CNC program easier to read. They are seldom required for CAD/CAM generated programs with no subprograms. Because they take up controlmemorymost 3D programs do not use block numbers. Block numbers are integers up to five characters long with no decimal point. They cannot appear before the tape start/end character (%) and usually do not appear before a comment only block. N100 T02 M06
O	Program name	Programs are stored on the control by their program number. Thisis an integer thatis preceded by the letter O and has no decimal places. O1234 (Exercise 1)

P	Serves as parameter address for various G and M codes	Dwell (delay) in seconds.Accompanied by G4 unless used within certain drill cycles. G4 P.1
Q	Peck increment in canned cycles	The incremental feed distance per pass in a peck drill cycle. G83 X2.000 Y2.000 Z-.625 F20.R.2 Q.2 P9.
R	Defines size of arc radius or defines retract height in canned cycles	Arcs can be defined using the arc radius R or I,J,Kvectors. IJK's are more reliable than R's so itis recommendedto use them instead. Ris also usedby drill cycles as the return plane Z value. G83 Z-.625 F20.R.2 Q.2 P9.
S	Defines speed, either spindle speed or surface speed depending on mode	Spindle speed in revolutions per minute (RPM). It is an integer value with no decimal, and always used in conjunction with M03 (Spindle on CW) or M04 (Spindle on CCW). S2500M03
T	Tool selection	Selects tool. It is an integer value always accompanied by M6 (tool change code). T01 M06
U	Incremental axis corresponding to X axis (typically only lathe group A controls) Also defines dwell time on some machines.	In these controls, X and U obviate G90 and G91, respectively. On these lathes, G90 is instead a fixed cycle address for roughing.
V	Incremental axis corresponding to Y axis	Until the 2000s, the V address was very rarely used, because most lathes that used U andWdidn'thave a Y-axis, so they didn't use V. (Green et al 1996 did not even list V in their table of addresses.) That is still often the case, although the proliferation of live lathe tooling and turn-mill machining has made V address usage less rare than it used to be (Smid2008 shows an example).
W	Incremental axis corresponding to Z axis (typically only lathe group A controls)	In these controls, Z and W obviate G90 and G91, respectively. On these lathes, G90 is instead a fixed cycle address for roughing.
X	Absolute or incremental position ofX axis.	Coordinate data for the X-axis. Up to four places after the decimalare allowedand trailing zeros are not used. Coordinates are modal, so there is no need to repeat them in subsequent blocks if they do not change. G01 X2.250F20.

Y	Absolute or incremental position of Y axis	Coordinate data for the Y-axis. G01 Y2.250 F20.
Z	Absolute or incremental position of Z axis	Coordinate data for the Z-axis.

Special Character Code Definitions

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

% – Program Start or End

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

( ) – Comments

Comments to the operator must be all caps and enclosed within brackets. The maximum length of a comment is 40 characters and all characters are capitalized.
(T02: 5/8 END MILL)

/ – Block Delete

Codes after this character are ignored if the Block Delete switch on the control is on.
/ M00

; – End of Block

This character is not visible when the CNC program is read in a text editor (carriage return), but does appear at the end of every block of code when the program is displayed on the machine control.
N8 Z0.750 ;

G & M Codes

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

G-Codes

Codes that begin with G are called preparatory words because they prepare the machine for a certain type of motion.

Table 3: G-Code

Code	Description
G00	Rapid motion.Used to position the machine for non-milling moves.
G01	Line motion at a specified feed rate.
G02	Clockwise arc.
G03	Counterclockwise arc.
G04	Dwell.
G28	Return to machine home position.
G40	Cutter Diameter Compensation (CDC) off.
G41	Cutter Diameter Compensation (CDC) left.
G42	Cutter Diameter Compensation (CDC) right.
G43	Tool length offset (TLO).
G54	Fixture Offset #1.
G55	Fixture Offset #2.
G56	Fixture Offset #3.
G57	Fixture Offset #4.
G58	Fixture Offset #5.
G59	Fixture Offset #6.
G80	Cancel drill cycle.
G81	Simple drill cycle.
G82	Simple drill cycle with dwell.
G83	Peck drill cycle.
G84	Tap cycle.

G90	Absolute coordinate programming mode.
G91	Incremental coordinate programming mode.
G98	Drill cycle return to Initial point (R).
G99	Drill cycle return to Reference plane (last Z Height)

M-Codes

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

Table 4: M-Codes

Code	Description
M00	Program stop.Press Cycle Start button to continue.
M01	Optional stop.
M02	End of program.
M03	Spindle on Clockwise.
M04	Spindle on Counterclockwise.
M05	Spindle stop.
M06	Change tool.
M08	Coolant on.
M09	Coolant off.
M30	End program and press Cycle Start to run it again.

Select G-Code Definitions (Expanded)

G00 – Rapid Move

This code commands the machine to move as fast as it can to a specified point. It is always used with a coordinate position and is modal. Unlike G01, G00 does not coordinate the axes to move in a straight line. Rather, each axis moves at its maximum speed until it is satisfied. This results in motion as shown in Figure 18, below.

G00 X0. Y0.

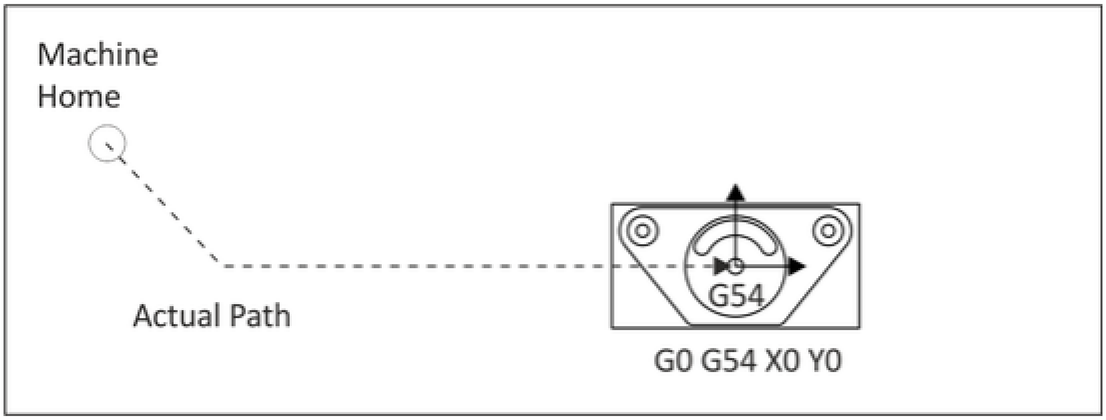


Figure 1. G00 Motion

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

Linear Motion is Straight Line Motion:

G-Code is about motion, and the most common kind of motion found in part programs is straight line or linear motion. Motion is another one of those things in G-Code that is modal. You tell the controller what kind of motion you'd like with a G-Code and it remembers to always make that kind of motion until you tell it to change using another G-Code.

G00 for Fast Positioning ; Rapids Motion as fast as your machine will go. Used to move the cutter through air to the next position it will be cutting.

G01 for Slower Cutting Motion; Feed Motion slower, for cutting. Feedrate set by "F" G-Code.

F-word = "F" as in "Feedrate".

S-word = "S" as in "Spindle Speed", address is rpm.

Specifying Linear Motion With X, Y, and Z:

Specifying G00 or G01 does not cause any motion to happen–they merely tell the controller what type of motion is expected when you finally tell it where to move to. For actual motion you need to specify a destination using the X, Y, and Z words. To move to the part zero, we might issue a command like this:

G00 X0 Y0 Z0   Or use G01 if you want to go slower   G01 X0 Y0 Z0 F40.

Interpolated motion or an interpolated move, When we specify multiple coordinates on a line, means more than one axis of the machine is moving at the same time. In fact, the controller will move them all at exactly the right speed relative to one another so that the cutter follows a straight line to the destination and moves at the feedrate.

If we specify the same destination, but spread the coordinates over multiple lines, each line is a separate move:

G00 X0 Y0 (Move to X0 Y0 in one move, keeping Z constant)

Z0 (Move to Z0 in one move, keeping X and Y constant)

G00 and G01 are modal, so we only have to specify them when we want to change modes.

Z Axis:

The concept of interpolated moves raises an interesting issue for the Z axis. It's often a good idea to move the depth-of-cut-axis on its own, rather than as coordinated motion with other axes(X and Y). Whether you're going to have a problem (collision) as the cutter gets close to the workpiece and fixturing. First moving in X and Y and then moving in Z , it's much easier to judge whether an accidental collision is about to take place. You're also much less likely to hit some random object sticking up, like a clamp, if you keep the cutter high until you're directly over where you want to start cutting.

G02 and G03 Circular Motion is a Mode Initiated:

G02 establishes a mode for clockwise circular arcs.

G03 establishes a mode for counter-clockwise circular arcs.

The G02 or G03 mode is established, arcs are defined in G-Code by identifying their 2 endpoints and the center which must be equi-distant from each endpoint. The endpoints are easy. The current control point, or location when the block is begun establishes one endpoint. The other may be established by XYZ coordinates. The center is most commonly identified by using I, J, or K to establish relative offsets from the starting point of the arc to the center.

EXAMPLE OF CLOCKWISE ARC:

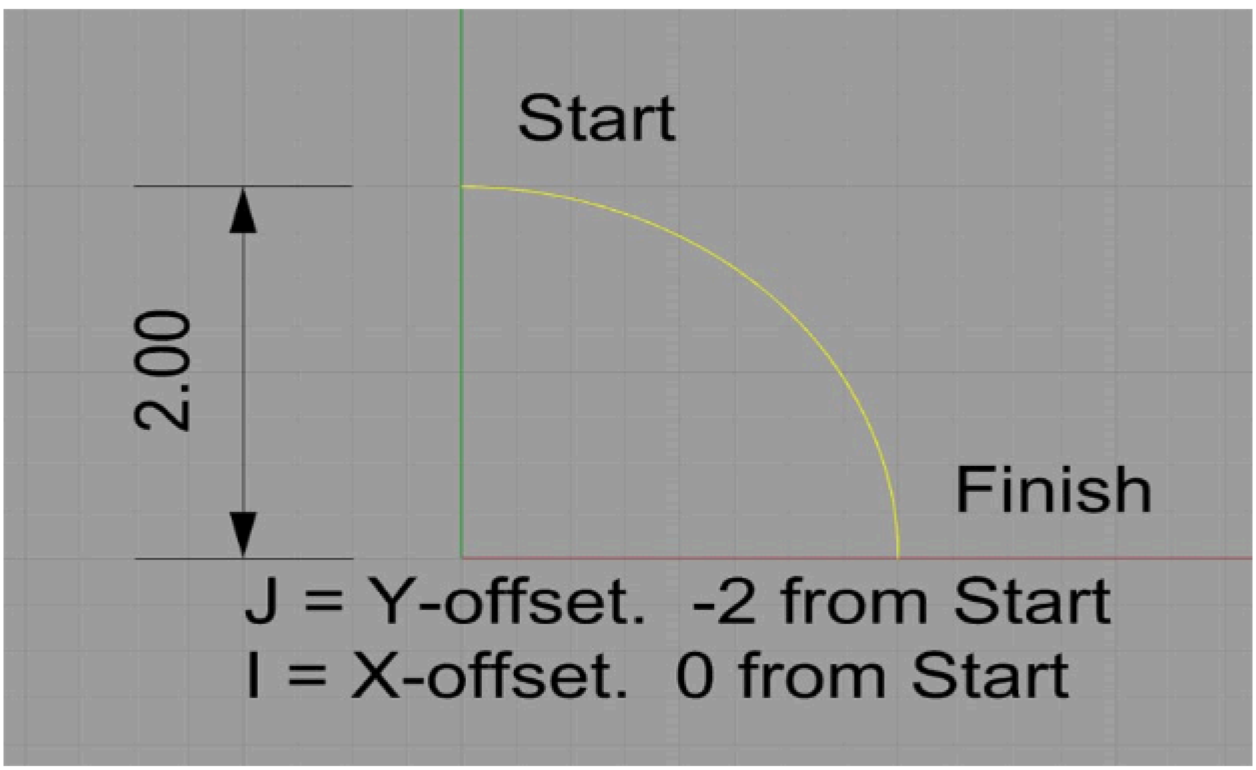


Figure 2. An Arc's center with IJK

This arc starts at X0Y2. and finishes at X2.Y0. It's center is at X0Y0. We could specify it in G-code like this:

G02 (Set up the clockwise arc mode)

X2Y0 I0J-2.0

The Center Using Radius "R".

The center just by specifying the radius of the circle. Circle has a radius of 2, so the G-Code might be simply:

G02 X2Y0 R2

G17/G18/G19 – Plane Designation

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

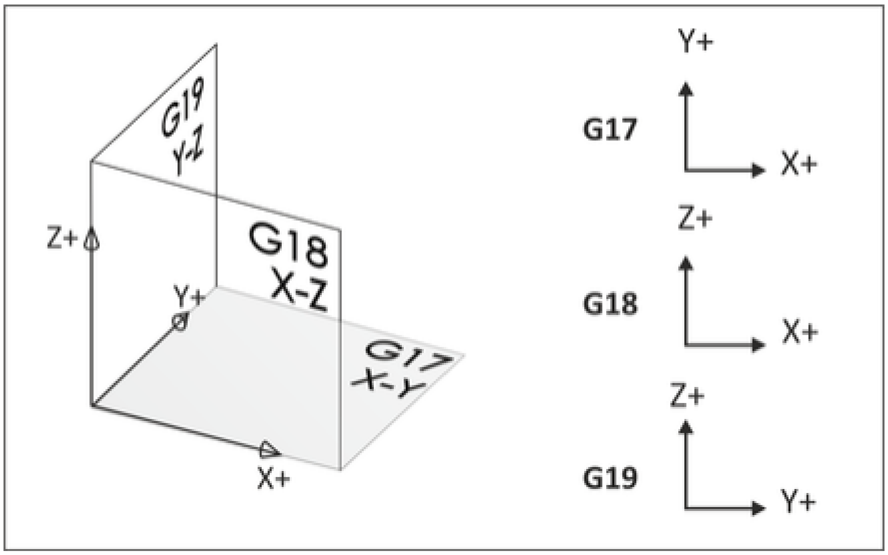


Figure 3. Plane Designation

G40/G41/G42 – Cutter Diameter Compensation (CDC)

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

Table 5: Diameter Offset Register

Tool Diameter Offset	Value
D1	0.0125
D2	0.0000
D3	0.0000
D4	0.0000
D5	0.0000
D6	0.0000

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

G01 G41 D1 X1.0 Y.25 F40.

G43 – Tool Length Compensation

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

The (TLO) can be thought of like a table on the control:
Table 6: Work Offsets

Tool Length Resister	Z
H1	10.236
H2	4.7510
H3	6.9652
H4	7.6841
H5	12.4483
H6	8.2250

The TLO is combined with the active fixture offset on the control so the machine knows where the tip of the tool is in relation to the part datum.

G43 H1 Z1.

G54 – Work Offset

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

Table 7: Work Offset

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

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

The X and Y values represent the distance from the machine home to part datum XY. The Z value is the distance from the tool reference point (for example, the top of a 1-2-3 block) and the part Z-datum.

G54 X0. Y0.

UNIT TEST

1. Please describe the CNC program list instruction.
2. All CNC program start and end with what?
3. Describe letter address Commands codes.
4. Please lists three special character codes.
5. Describe G and M codes.
6. Please describe G00 G90 G54 X0 Y0.
7. Please describe G00 G90 G43 H1 Z1.
8. What is the different between G00 and G01?
9. Explain the different between G02 and G03.
10. Please Describe the F and S word.

22

Unit 5: CNC Operation

OBJECTIVE

After completing this unit, you should be able to:

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

Overview of CNC Setup and Operation

CNC machine setup and operation follows the process below:

1. Pre-Start
2. Start/Home
3. Load Tools
4. Mount Remove Part into the vise
5. Set Tool Length Offsets Z
6. Set Part Offset XY
7. Load CNC Program
8. Dry Run
9. Run Program
10. Adjust Offsets as Needed
11. Shut Down

1. Pre-Start

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

2. Start/Home

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

3. Load Tools

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

4. Mount the Part in the Vise

Place the Part to be machine in the vise and tighten.

5. Set Tool Length Offsets

Set Tool Length Offsets For each tool used in the order listed in the CNC program, jog the Tools to the top of the part and then set the TLO.

6. Set Part Offset XY

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

7. Load CNC Program

Load your CNC program into CNC machine control using USB flash memory, or floppy disk.

8. Dry Run

Run the program in the air about 2.00 in. above the part .

9. Run Program

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

10. Adjust Offsets as Required

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

11. Shut Down

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

UNIT TEST

1. Please list the CNC setup and operation process steps.

2. Describe each process.

23

Unit 6: Haas Control

OBJECTIVE

After completing this unit, you should be able to:

- Identify the Haas Control.
- Identify the Keyboard.
- Describe Start/Home Machine procedure.
- Describe Door Override procedure.
- Describe Load Tools procedure.
- Describe Tool Length Offset (TLO) for each tool.
- Verify part zero offset(XY) using MDI.
- Describe the setting tool offset.
- Verify Tool Length offset using MDI.
- Describe the procedure of load CNC program.
- Describe the procedure of save CNC program.
- Explain how to run CNC program.
- Describe the use of cutter diameter compensation.
- Describe the shut down program.

Haas Control

The Haas control is shown in Figures 18 and 19. Familiarize yourself with the location of buttons and controls. Detailed instructions on the following pages show how to operate the control.

Figure 1. Haas CNC Mill ControlHaas Keyboard

Keyboard

Keyboard keys are grouped into these functional areas:

1. Function Keys

2. Cursor Keys

3. Display Keys

4. Mode Keys

5. Numeric Keys

6. Alpha Keys

7. Jog Keys

8. Overrides Keys

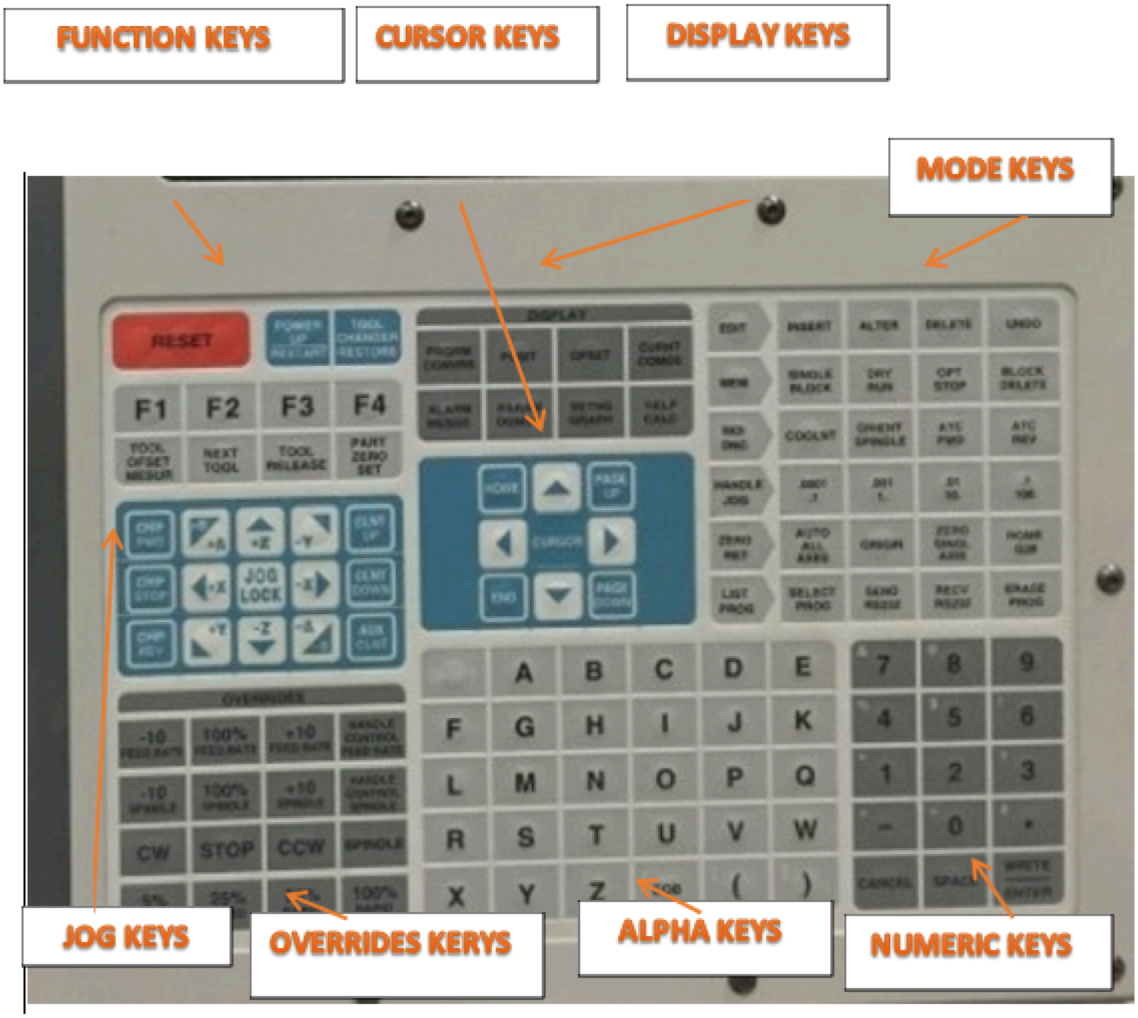


Figure 2. Haas CNC Control Buttons/ Keyboard keys

Start/Home Machine

Checklist:

1. Work Area: Made sure work area is Clear
2. Main Breaker: Turn On
3. Air Supply: Turn On Air to Correct pressure (at least 70PSI for tool changer to operate)
4. POWER ON: Press Green Button
5. Ensure Emergency Stop is not tripped. If it is, twist red knob right to release.
6. Wait until message 102 SERVOS OFF appears before proceeding.
7. RESET

8. Power On Restart

9. Ensure doors are closed and work area is clear.

10. Allow all machine axes to home before proceeding

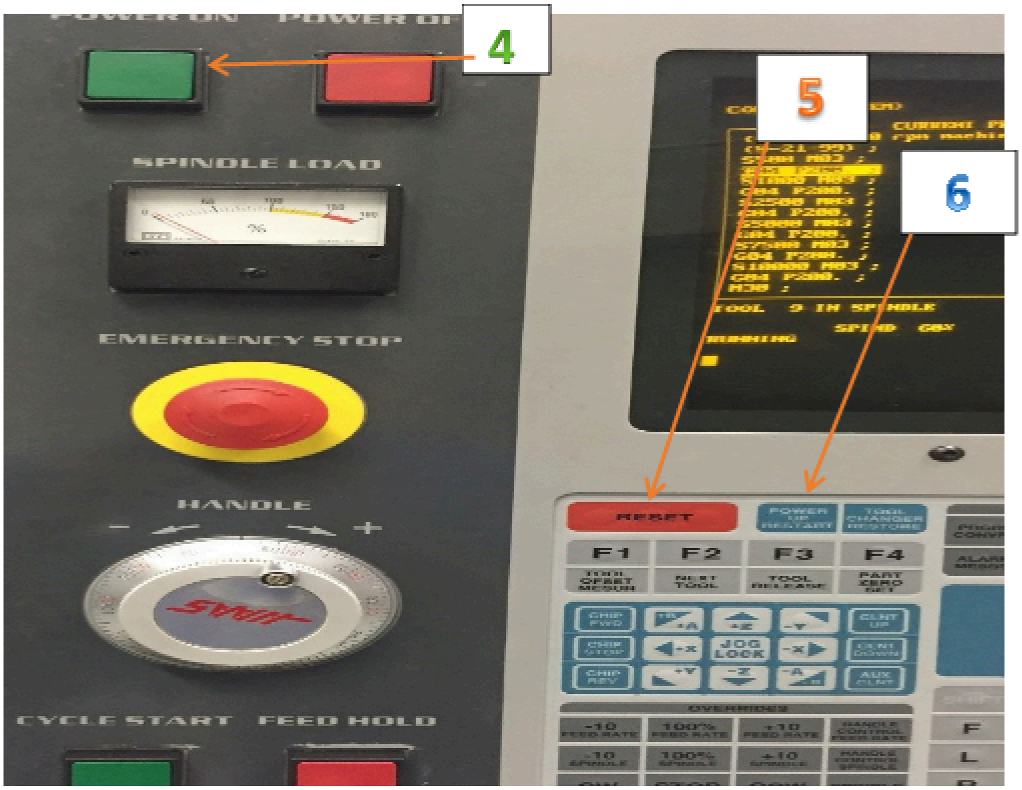


Figure 3. Start/Home Machine

Door Override

1. Mem: Select and Press Mem.

2. Setting Graph: Select and Press Setting Graph

3. Enter 51

4. Cursor: Press down arrow key and then the right arrow key to turn off

5. Write/Enter: Select and Press Write/Enter

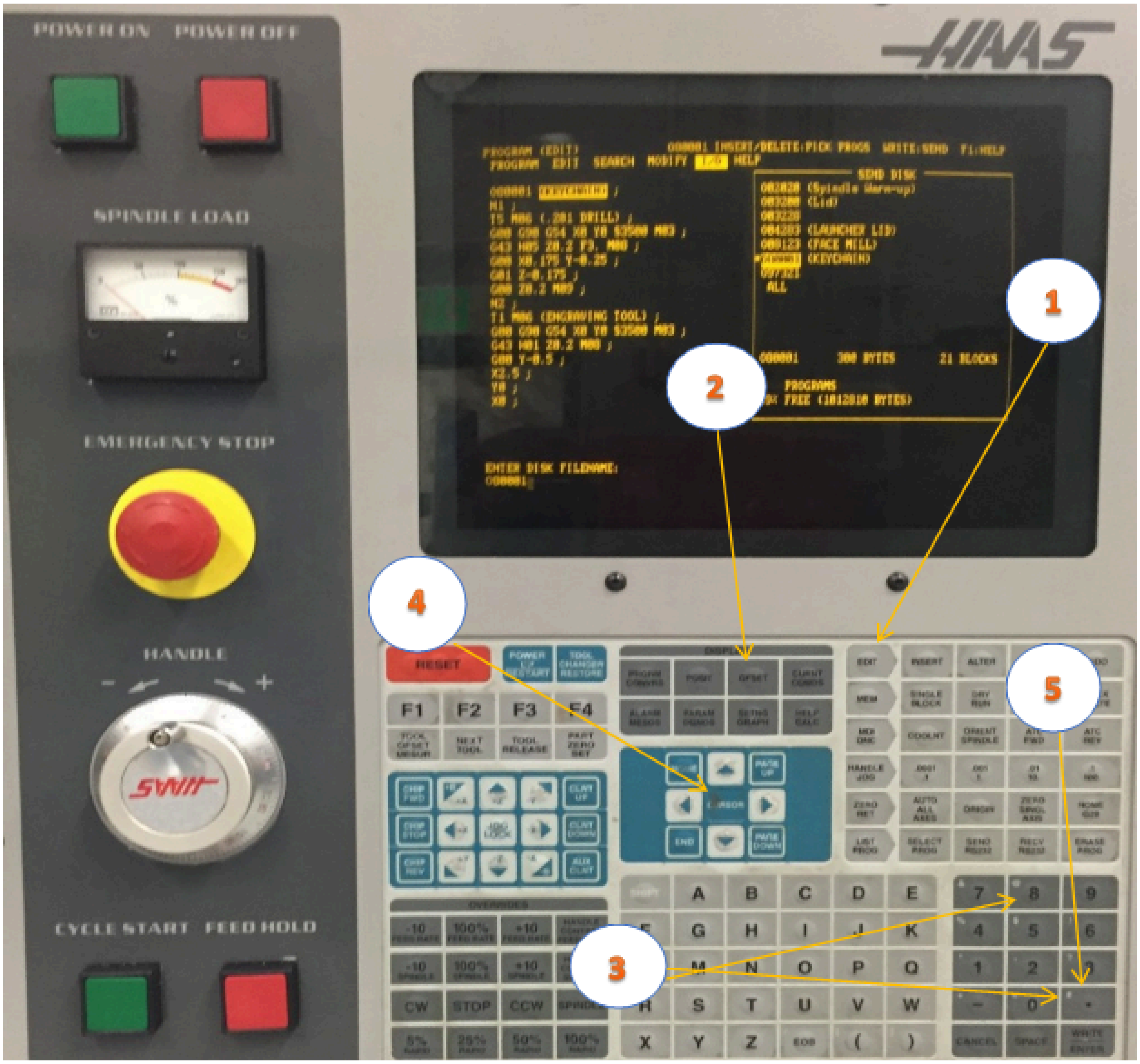


Figure 4. Door Override

Load Tools

Checklist:

1. MDI/DNC Key: Press the MDI/DNC button.

2. Tool Number:

- For example, to position the tool changer to T1,
- Press the T and then the 1 buttons.

3. ATC FWD: Press the ATC FWD button.

- Tool carousel will index to T1 position.

4. Position Tool in Spindle

- Do not grip by tool cutting flutes!
- Ensure tool taper is clean.
- Grip tool holder below V-flange to prevent pinching.
- Push tool into spindle.
- Ensure "dogs" on spindle line up with slots on tool holder.

5. Tool Release: Press the Tool Release button

- Machine will blow air thru spindle to clear debris.
- Gently push the tool upward and then release the Tool Release button.
- Ensure tool is securely gripped by spindle before releasing it.

6. Repeat steps 2-5 until all tools are loaded.

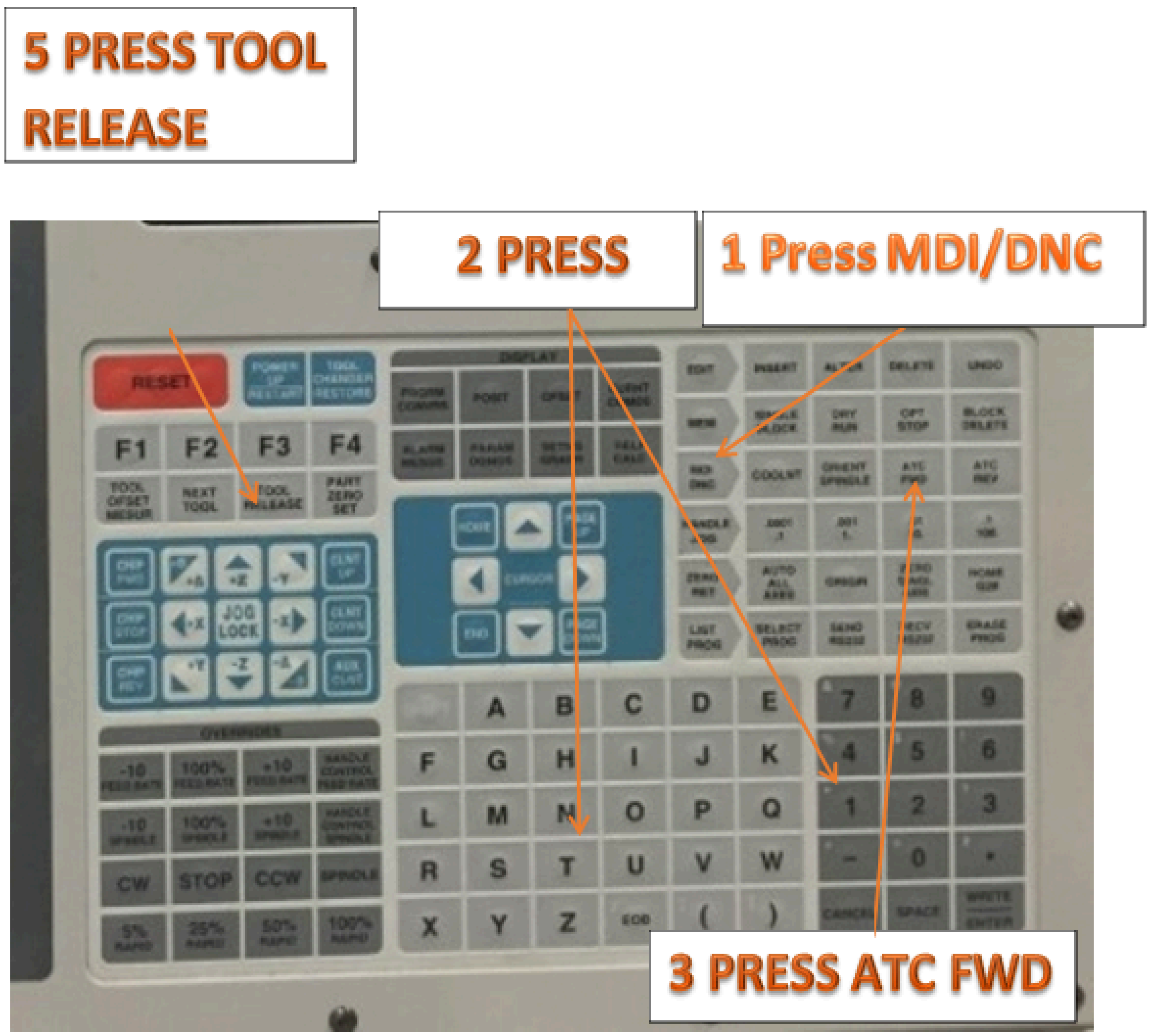


Figure 5. Load Tools

Setting Offsets

To machine a part accurately, the mill needs to know where the part is located on the table and the distance from the tip of the tools to the top of the part (tool offset from home position).

To manually enter offsets:

1. Choose one of the offsets pages.

2. Move the cursor to the desired column.

3. Type the offset value you want to use.

4. Press (ENTER) or (F1). The value is entered into the column.

5. Enter a positive or negative value and press (ENTER) to add the amount entered to the number in the selected column; press (F1) to replace the number in the column.

Jog Mode

Jog mode lets you jog the machine axes to a desired location. Before you can jog an axis, the machine must establish its home position. The control does this at machine power-up.

To enter jog mode:

1. Press (HANDLE JOG).

2. Press the desired axis (+X, -X, +Y,-Y, +Z, -Z).

3. There are different increment speeds that can be used while in jog mode; they are (.0001), (.001), (.01) and (.1). Each click of the jog handle moves the axis the distance defined by the current jog rate. You can also use an optional Remote Jog Handle (RJH) to jog the axes.

4. Press and hold the handle jog buttons or use the jog handle control to move the axis.

Set Tool Length Offset (TLO)

Checklist:

1. Handle Jog Mode: Select the Handle Jog button.

    - This sets machine to be controlled by the hand wheel.

2. Jog Increment: .01

    - This sets the job increment so each click of the hand wheel moves the tool .01 inches in the jog direction.

3.Jog Direction: Press the Z button

    - This sets the tool to move in Z when the jog handle is moved.

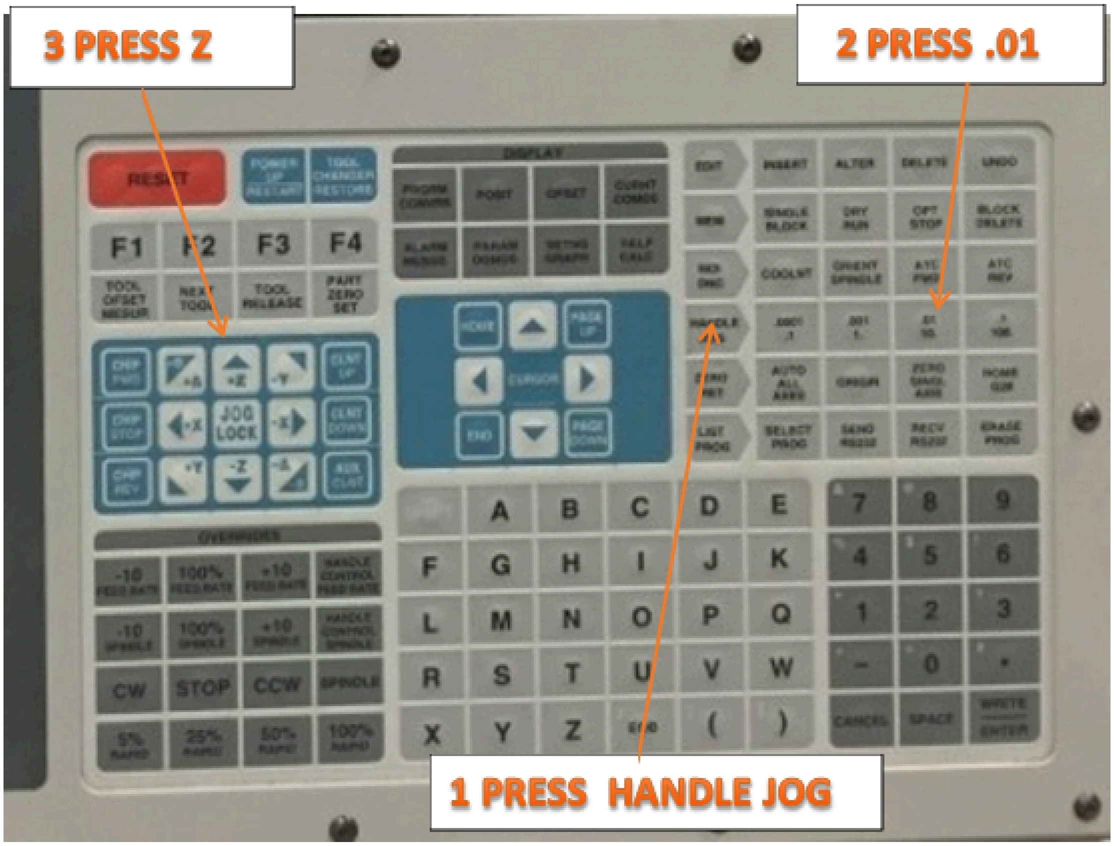


Figure 6. TLO

4. Offsets: Select and Press Offset

- Tool Offset page displays.

5. Cursor Arrows: Align for active tool

- Use the Up-Dn cursor keys (if needed) to move the highlighted bar on the graphics display over the offset values for the currently active tool.

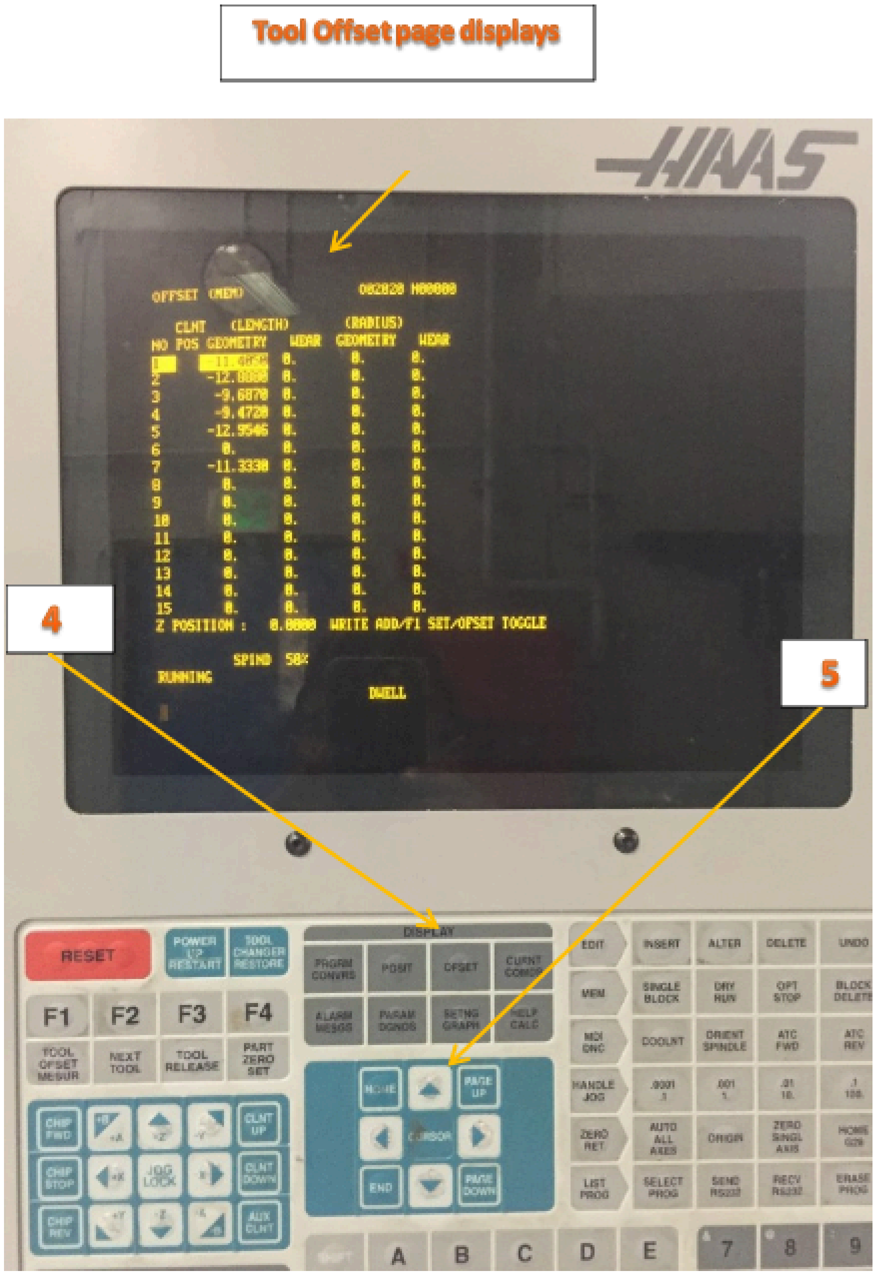


Figure 7. Offset

6.Use 1-2-3 Block to Set Tool Length

- Jog so tool is below top block.
- Apply slight pressure to block against tool. Use Jog Wheel to raise tool until the block just slides underneath it.
- Move block out of way and then move tool back down .01 inches below top of block.

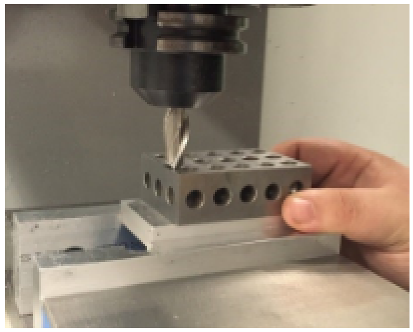


Figure 8

7. Jog Increment: .001

- Reduce jog increment and use jog handle to raise tool in .001 increments until it just slides under the block again.

8. Tool Offset Measure: Select and Press Tool Offset Measure

- This causes the control to enter the current position of the tool in the length offset register.
- Make sure the tool length number updates before proceeding.

9. Next Tool: Select and Press Next Tool

- This causes the current tool to be put away and the next tool to be loaded.
- Repeat steps 1 thru 9 until all tools are set.

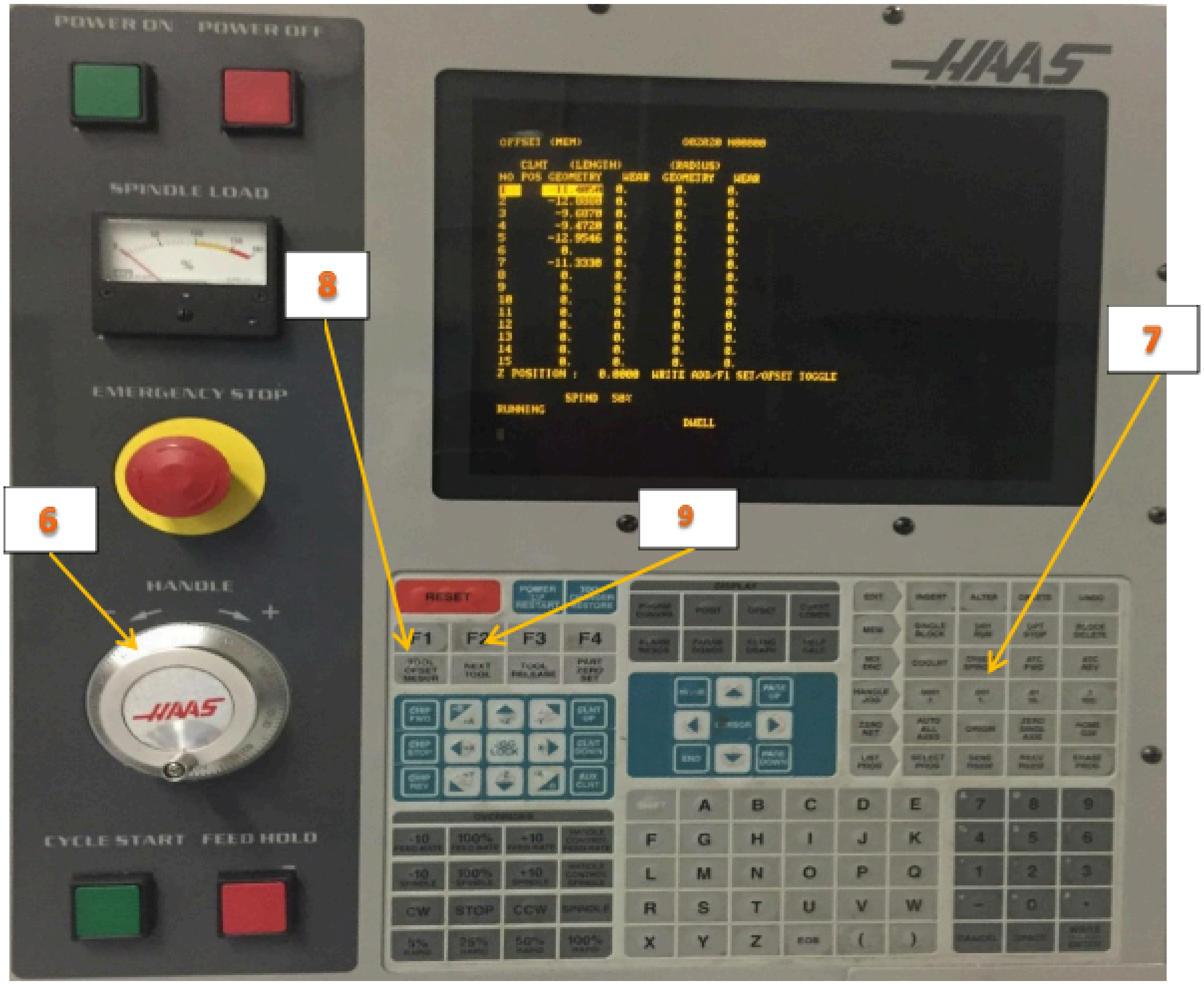


Figure 9. Offset Display Page

NOTE:

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

• The spindle must be off.

• Never place your hand between the tool and the workpiece.

• Ensure the correct axis and jog increment are set before jogging.

• Move the handle slowly and deliberately. Keep your eyes on your hands and the tool position at all times.

• Never allow anyone else to operate the control when your hand is in the work area.

How to Access MDI

The Manual Data Input Mode (MDI) is one of the modes your CNC machine can operate in. The idea is to enter G-Codes or M-Codes on a line which are executed immediately by the machine–you don't have to write an entire g-code program when a line or two will suffice. MDI offers a lot of power while requiring very little learning. You can even use MDI commands to machine your part. With MDI, CNC can be quick and dirty just like manual machining.

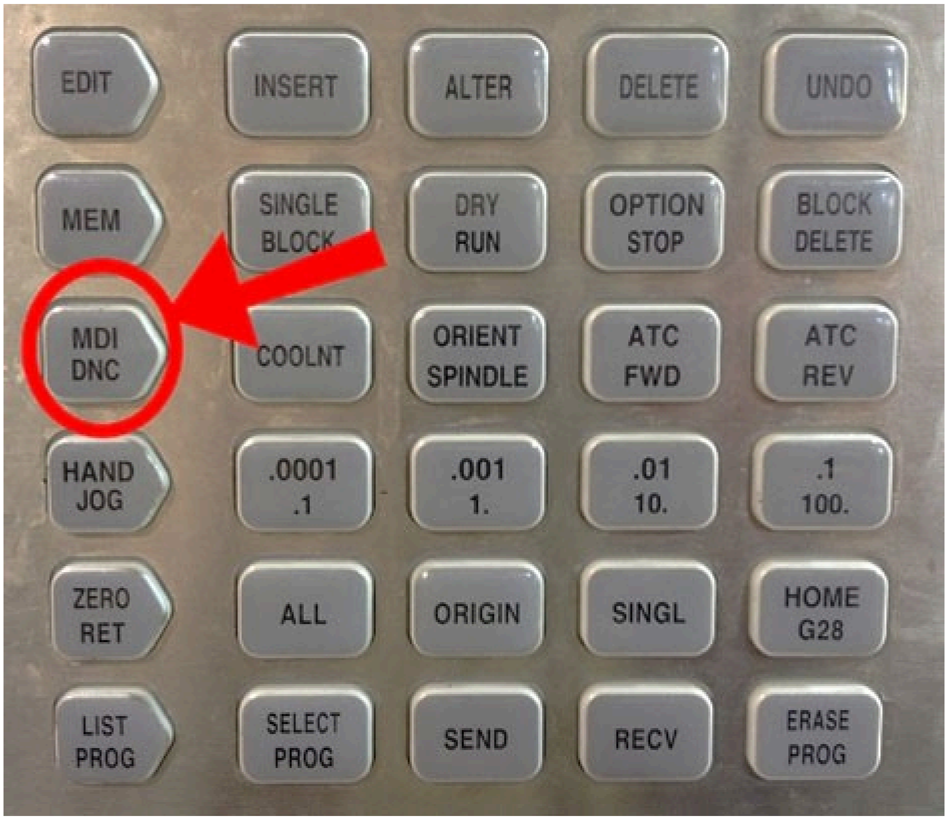


Figure 10. MDI

Press the MDI key on your CNC control panel to go to MDI mode.

For Example:

Press MDI/DNC

Erase Prog: Select and Press (to clear any commands)

Enter S1200 M03 (Spindle Speed 1200 RPM, On CW)

Setting Part Zero Offset OffsetXY(Using Edge Finder)

Checklist:

1. MDI/DNC Key: Select and Press MDI/DNC

2. Erase Prog: Select and Press Erase Prog to clear any commands

3. Turn on Spindle Speed: S1200

   - Press S1200 M03(Input): Select: Write/Enter

4. Cycle Start: Select

- Spindle will start CW at 1200 RPM

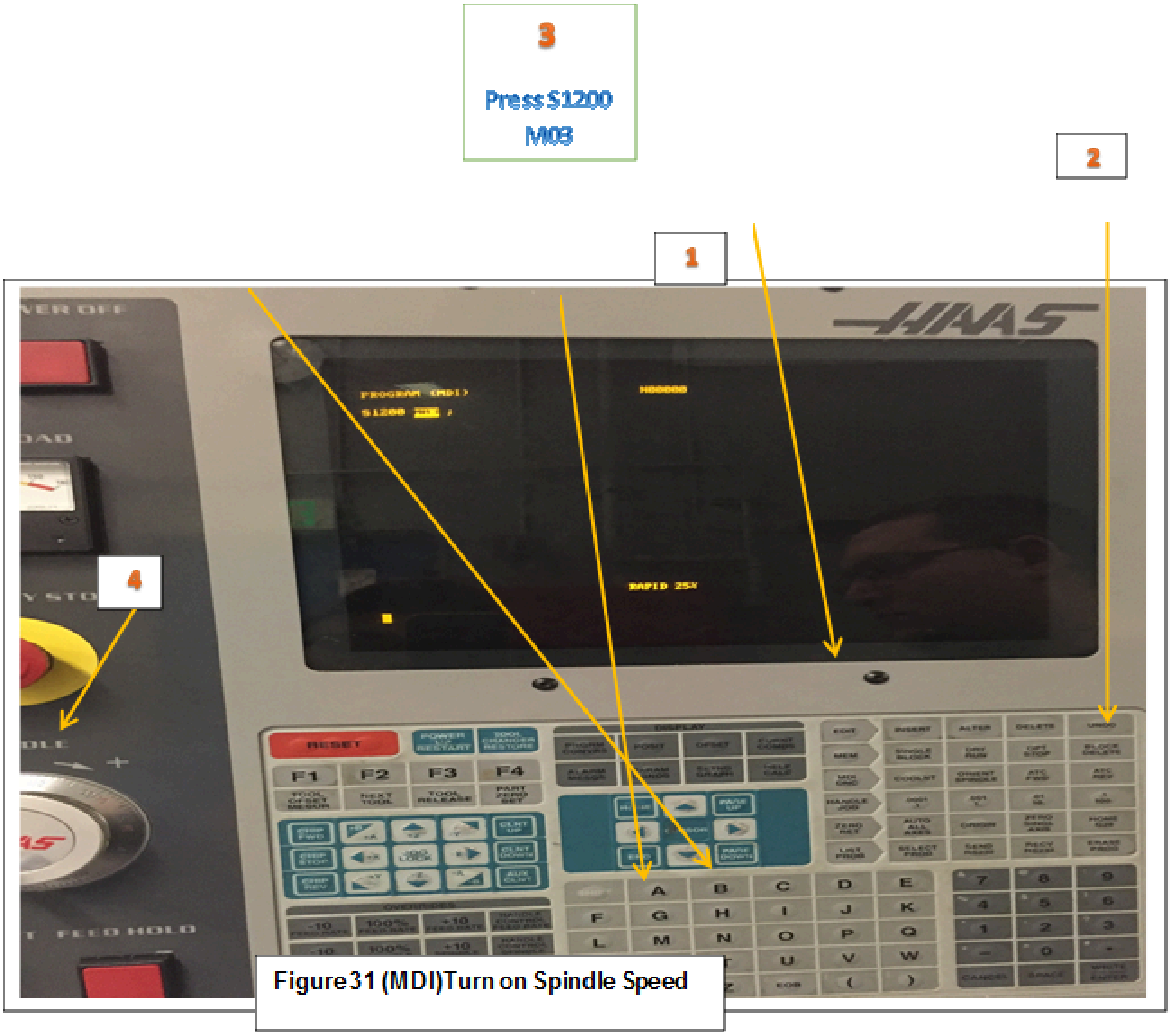
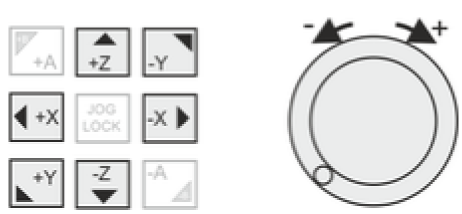


Figure 11. Turn on Spindle Speed (MDI)

5. Handle Jog: Select Handle Jog and Jog Increment: .01

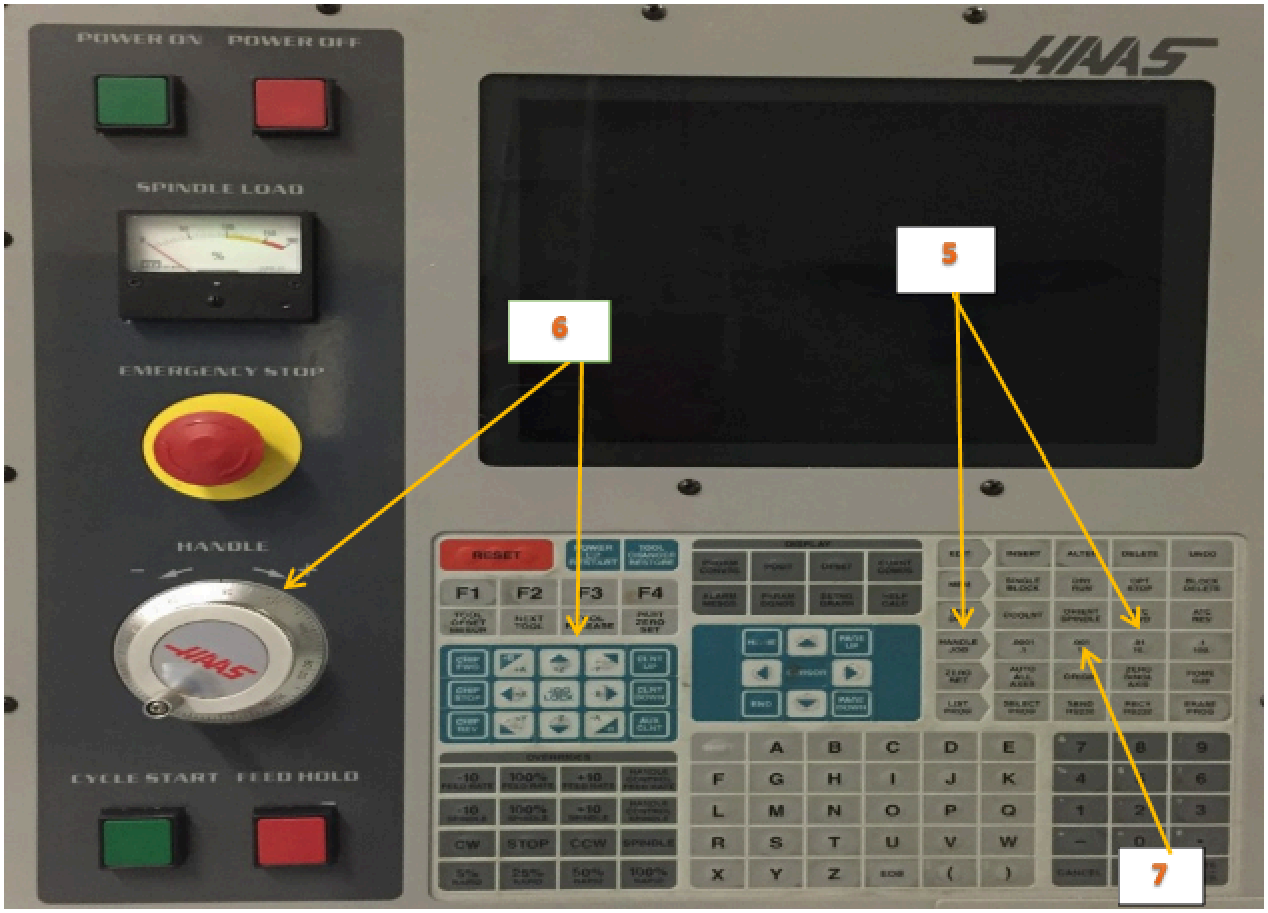
6. Jog Handle: As Needed

- Select jog direction and use handle as required to place edge finder stylus alongside the left part edge.



7. Jog Increment: .001

- Move edge finder slowly until it just trips off center as shown below.
- This places the center of the spindle exactly .100 from the part edge



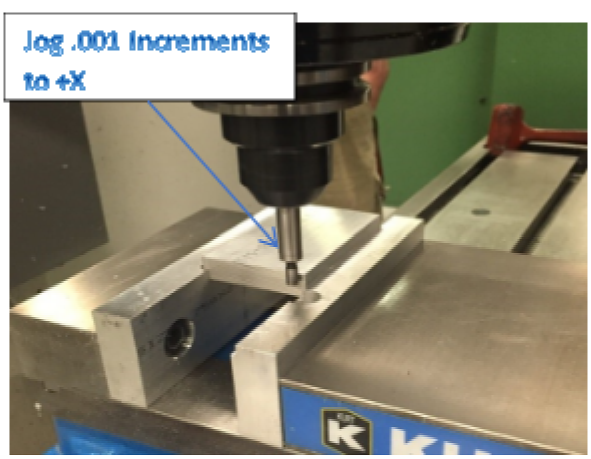


Figure 12. Just before tripping

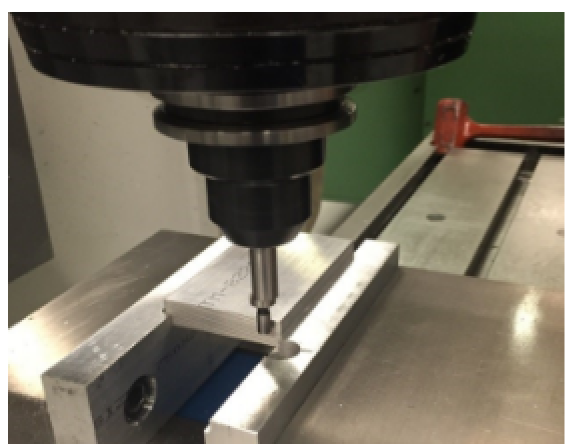
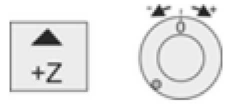


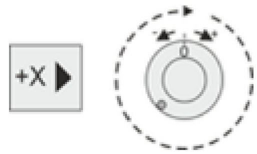
Figure 13. Just after tripping

8 .Jog Handle: Retract in Z



• Jog straight upward in Z until edge finder is above part and jog handle reads zero on the dial.

9. Jog Handle: Set jog direction to +X and rotate handle one full turn clockwise.



Since the control is in .001 increment mode, rotating the dial exactly one full turn places the center of the spindle directly over the left part edge.

10. Offset Page: Select and Press

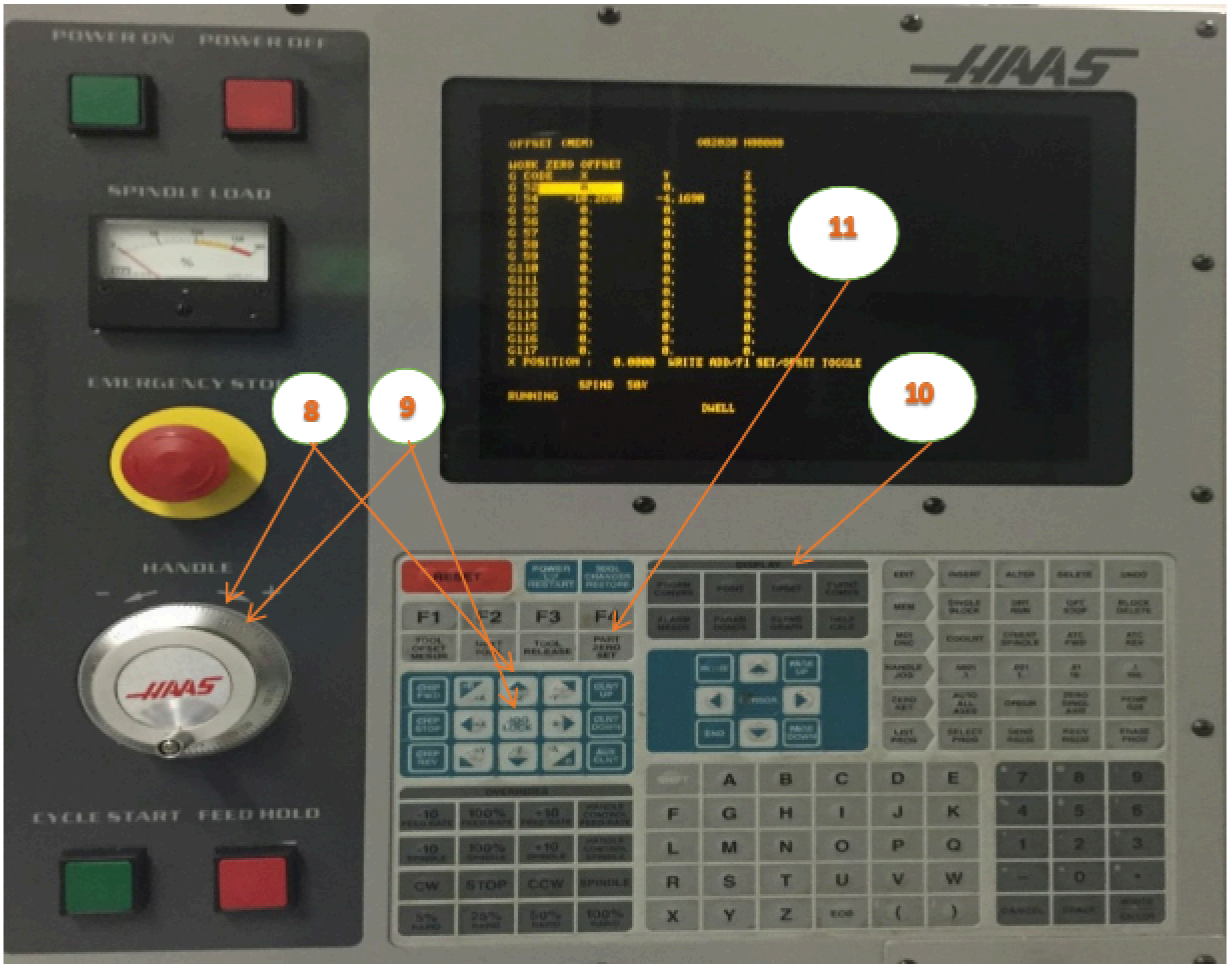
• Select Offset button and PgUp/PgDn buttons until Work Zero Offset page appears. Use Arrow keys to highlight G54 (or whatever fixture offset is to be set).

11. Part Zero Set: Press Part Zero Set

• This sets the G54 X value to the current spindle position.

12. Spindle Stop: Press spindle stop

13. For Setting the Y-axis repeat steps 6–11



Using MDI to verify Part zero offset

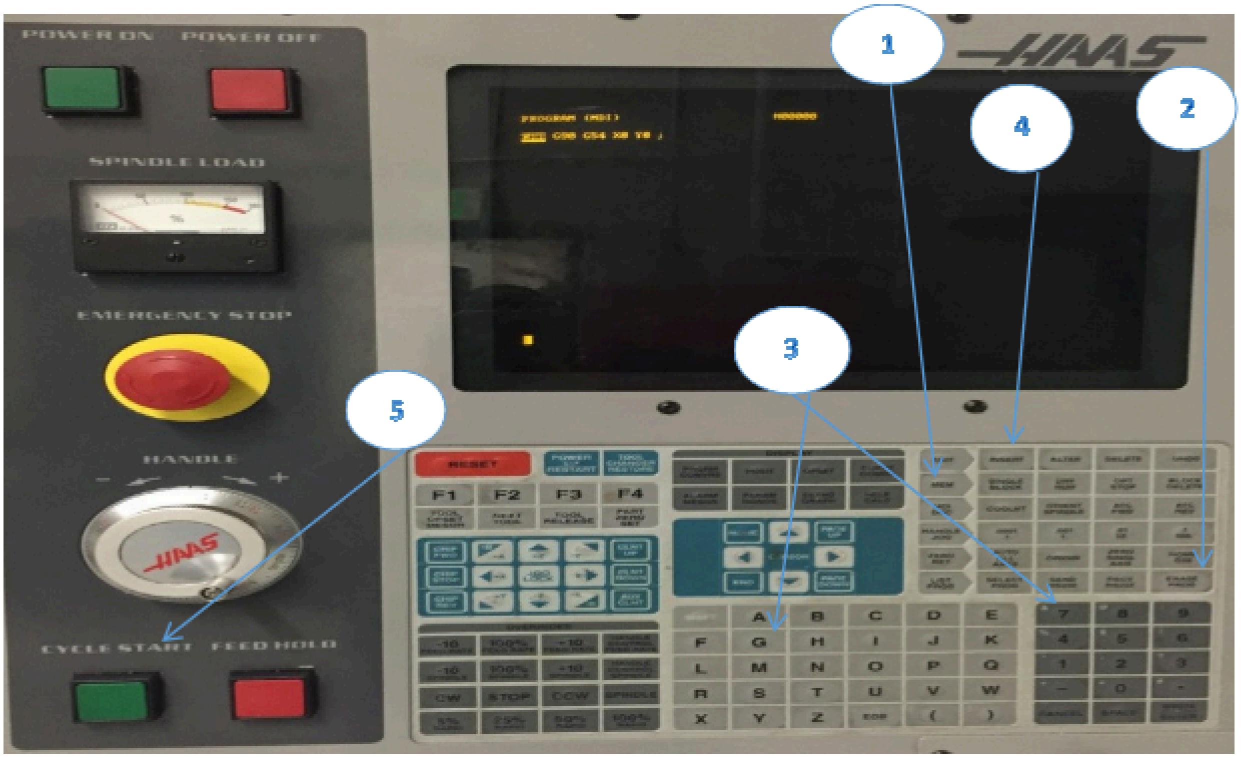
1. MDI/DNC Key: Select and Press MDI/DNC

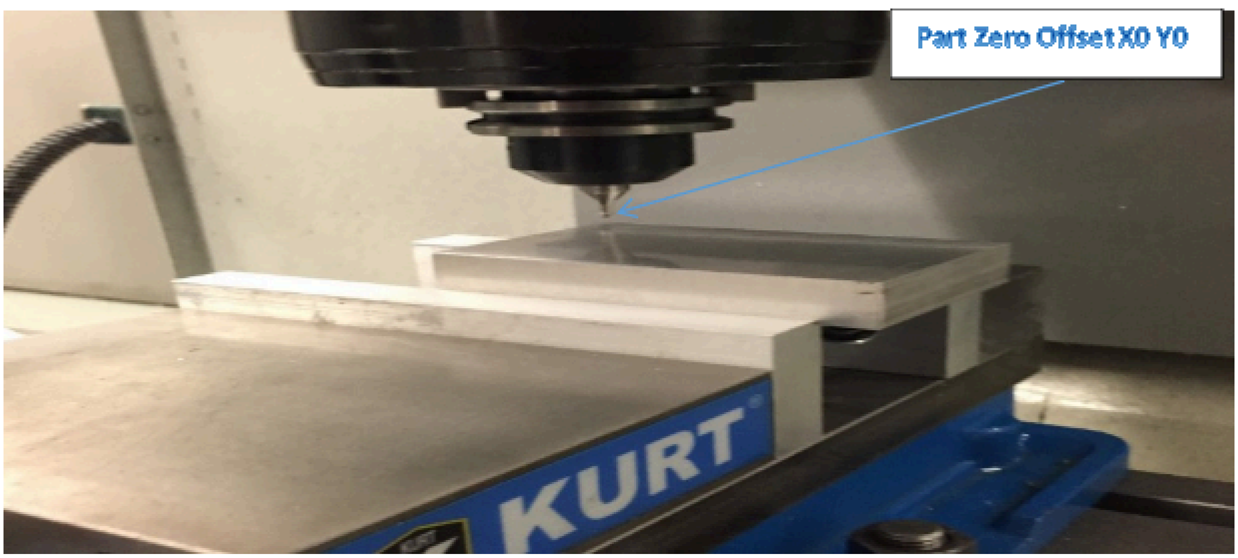
2. Erase Prog: Select and Press (to clear any commands)

3. Enter G00 G90 G54 X0 Y0

4. Insert: Select and Press Insert

5. Cycle Start: Select and Press Cycle Start





Part Zero Offset X0 Y0 (MDI)

- To shift the datum RIGHT in relation to the machine operator, ADD a shift amount to the offset X-value. For example, to shift X+.1, input .1 WRITE/ENTER.
- To shift the datum CLOSER to the machine operator, SUBTRACT a shift amount from the offset Y-value. For example, to shift Y-.1, input -.1 WRITE/ENTER

Setting Part Zero Offset XY(Using Mechanical pointer)

In order to machine a part, the mill needs to know where the part is located on

the table. You can use an edge finder, an electronic probe, or many other tools and methods to establish part zero. To set the part zero offset with a mechanical pointer:

1. Place the material (1) in the vise and tighten.

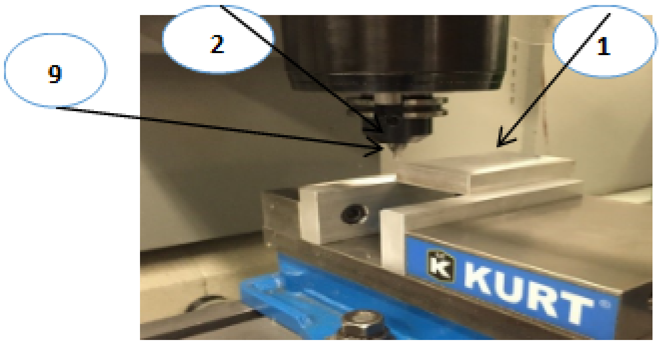


Figure 14. Setting Part Zero XY (Using Mechanical pointer)

2. Load a pointer tool (2) in the spindle.

3. Press (HANDLE JOG) (E).

4. Press (.1/100.) (F) (The mill will move at a fast

speed when the handle is turned).

5. Press (+Z) (A).

6. Handle jog (J) the Z-Axis approximately 1″ above the part.

7. Press (.001/1.) (G) (The mill will move at a slow speed when the handle is turned).

8. Handle jog (J)the Z-Axis approximately. 0.2″ above the part.

9. Select between the X and Y axes (I) and handle jog (J) the tool to the upper left corner

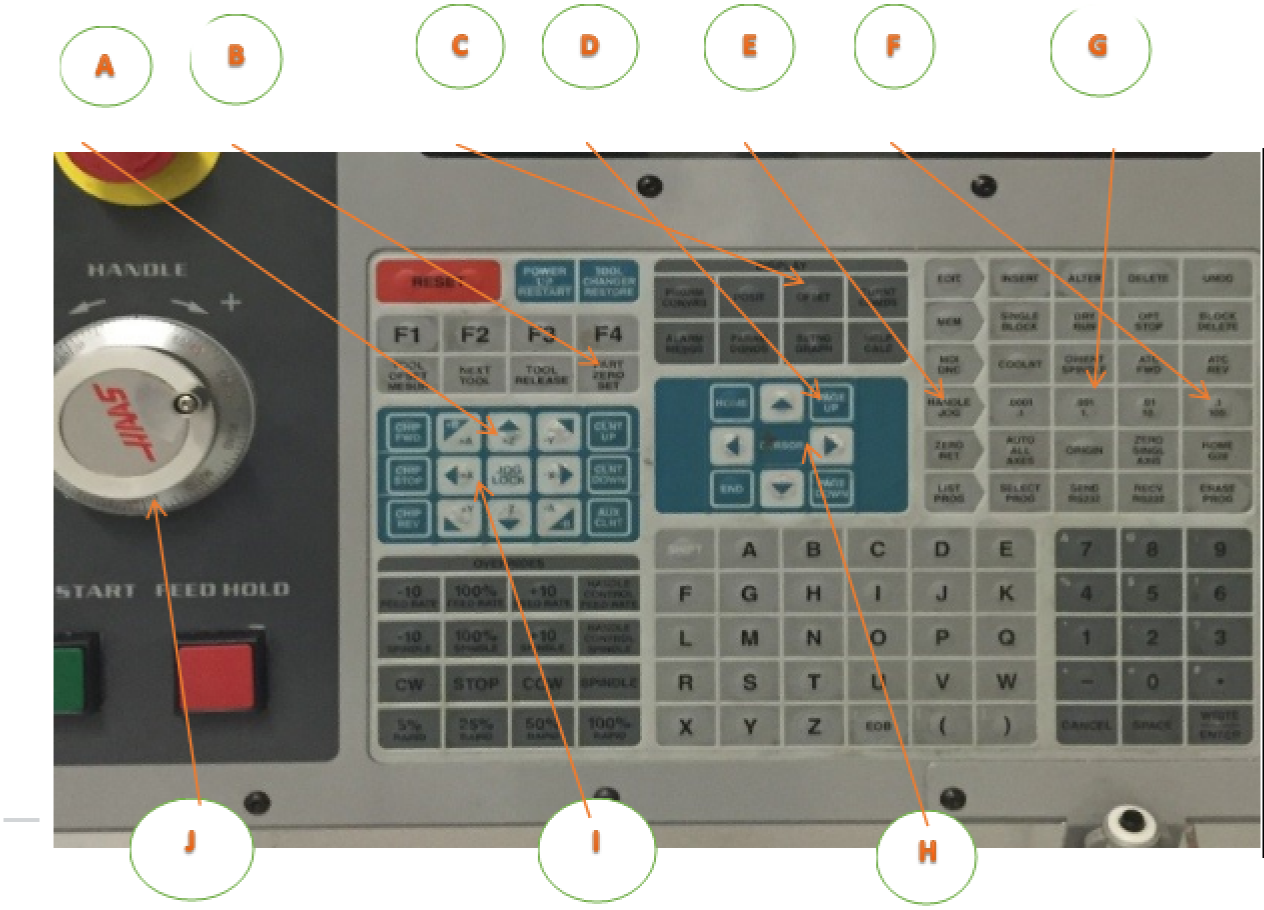
of the part (See Figure 36 above (9).

10. Press (OFFSET) (C) until the Active Work Offset pane is active.

11. Cursor (H) to G54 X-Axis column.

12. Press [PART ZERO SET] (B) to load the value into the X-Axis column. The second

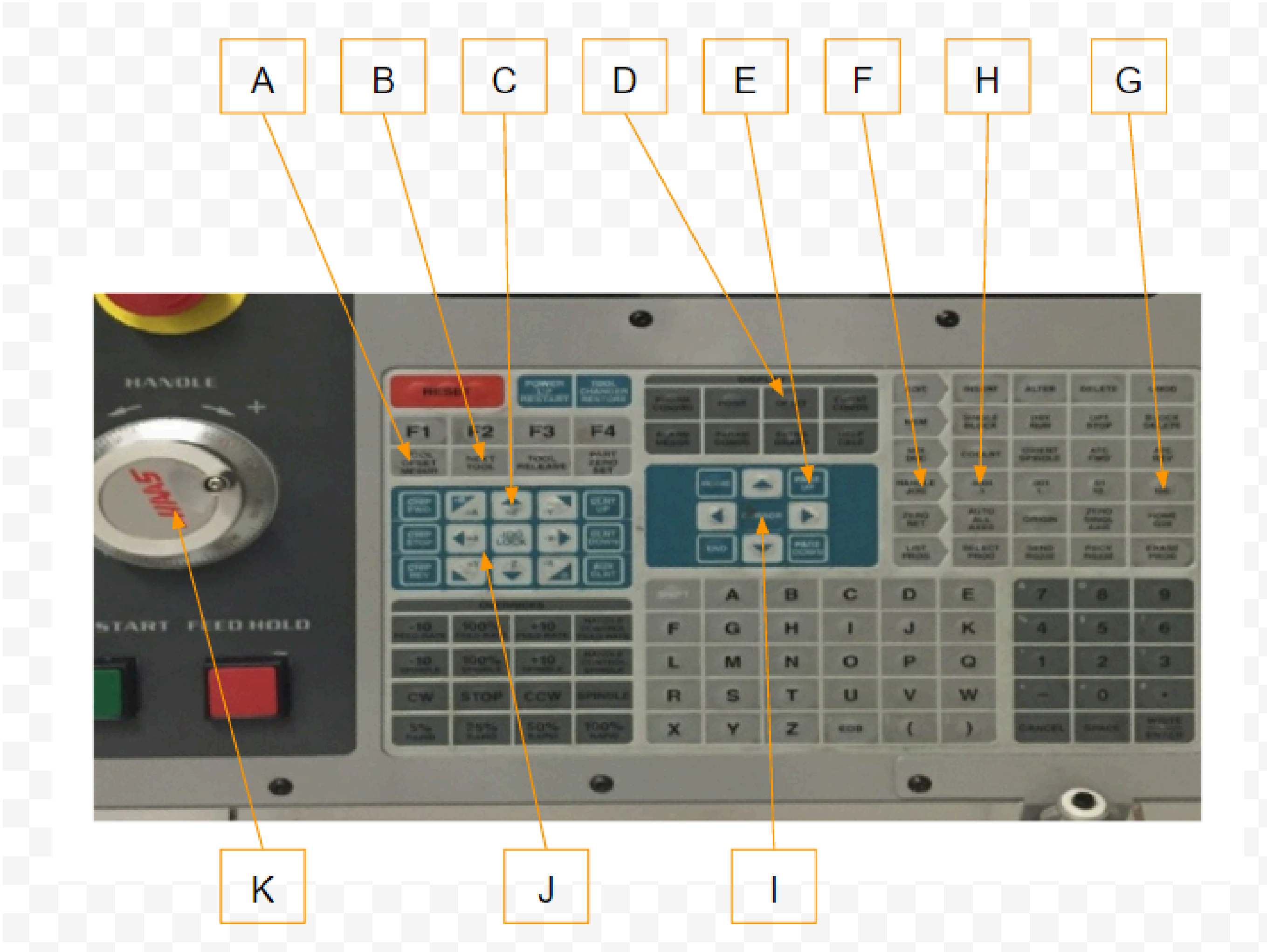
press of [PART ZERO SET] (B) loads the value into the Y-Axis column.



Setting Tool Offset

The next step is to touch off the tools. This defines the distance from the tip of the tool to the top of the part. Another name for this is Tool Length Offset, which is designated as H in a line of machine code. The distance for each tool is entered into the Tool Offset Table.

Setting Tool Offset. With the Z Axis at its home position, Tool Length Offset is measured from the tip of the tool (1) to the top of the part (2). See Figure 15.



1. Load the tool in the spindle (1).

2. Press (HANDLE JOG) (F).

3. Press (.1/100.)(G) (The mill moves at a fast rate when the handle is turned).

4. Select between the X and Y axes (J) and handle jog (K) the tool near the center of the part.

5. Press (+Z) (C).

6. Handle jog (K) the Z Axis approximately 1″ above the part.

7. Press (.0001/.1) (H) (The mill moves at a slow rate when the handle is turned).

8. Place a sheet of paper between the tool and the work piece. Carefully move the tool down to the top of the part, as close as possible, and still be able to move the paper.

9. Press (OFFSET) (D).

10. Press (PAGE UP) (E) until you display the Program Tool Offsets window. Scroll to tool #1.

11. Cursor (I])to Geometry for position #1.

12. Press [TOOL OFFSET MEASURE] (A).

The next step causes the spindle to move rapidly in the Z Axis.

13. Press (NEXT TOOL) (B).

14. Repeat the offset process for each tool.

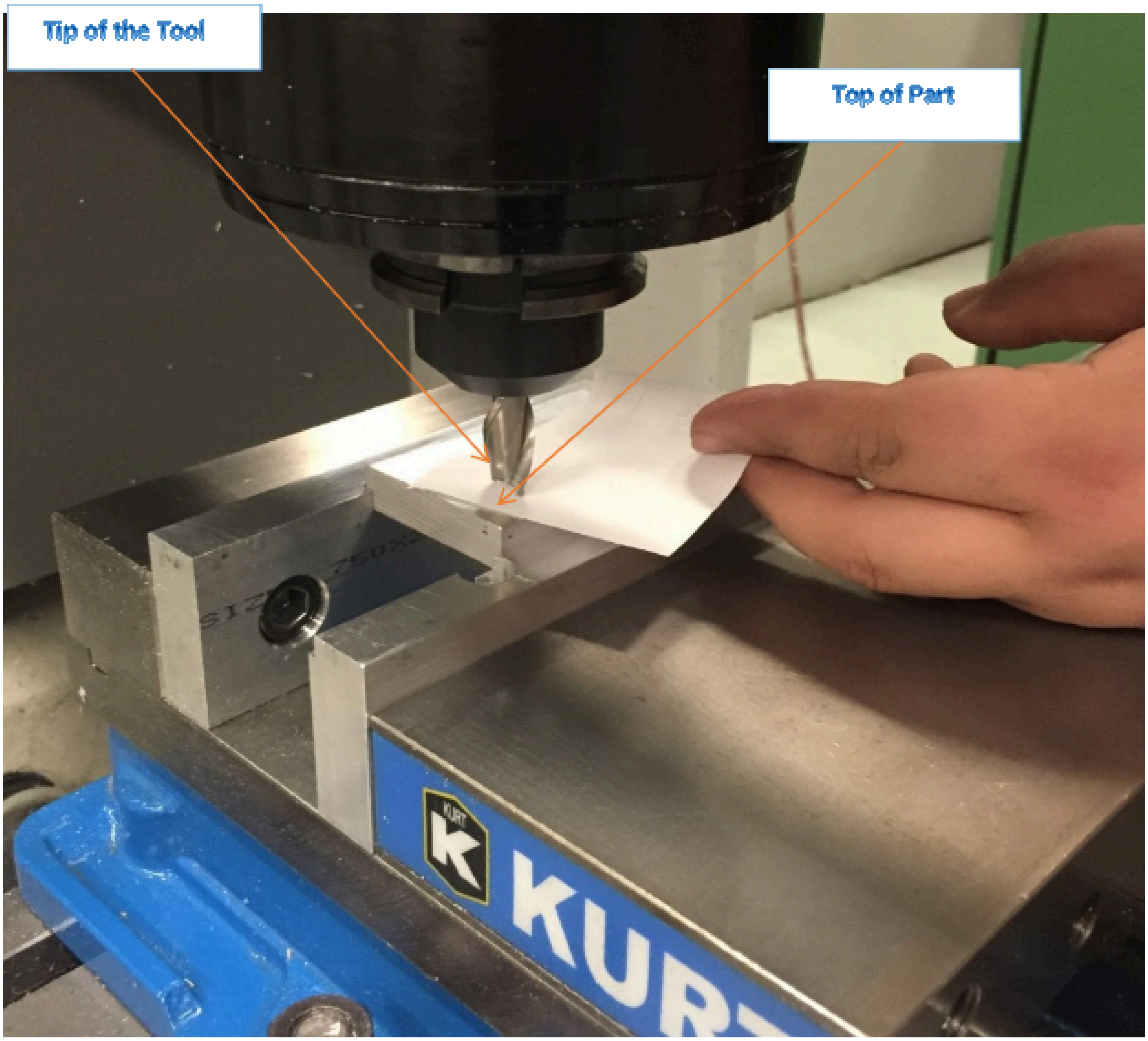


Figure 15. Setting Tool Offset (sheet of paper)

Using MDI to verify Tool Length offset

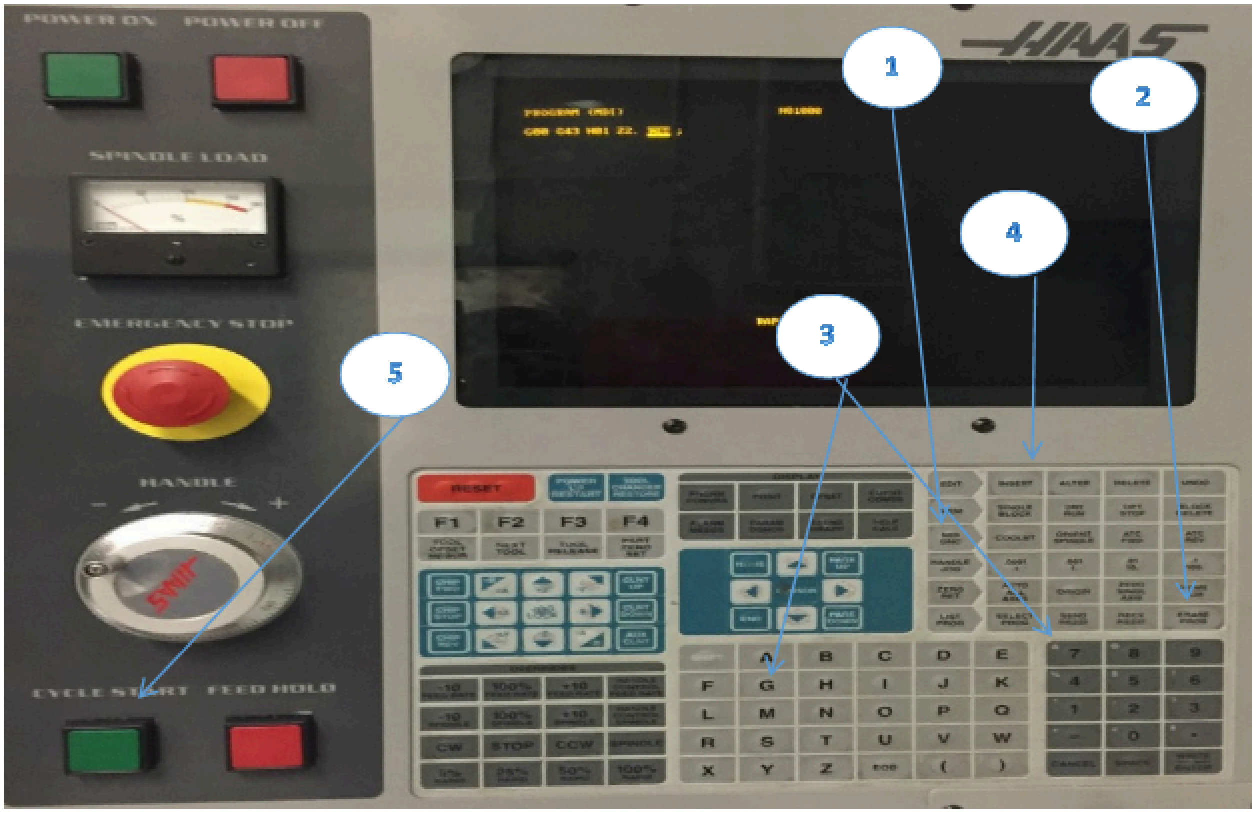
1. MDI/DNC Key: Select and Press MDI/DNC

2. Erase Prog: Select and Press (to clear any commands)

3. Enter G00 G90 G43 H01 Z2.00

4. Insert: Select and Press Insert

5. Cycle Start: Select and Press Cycle Start



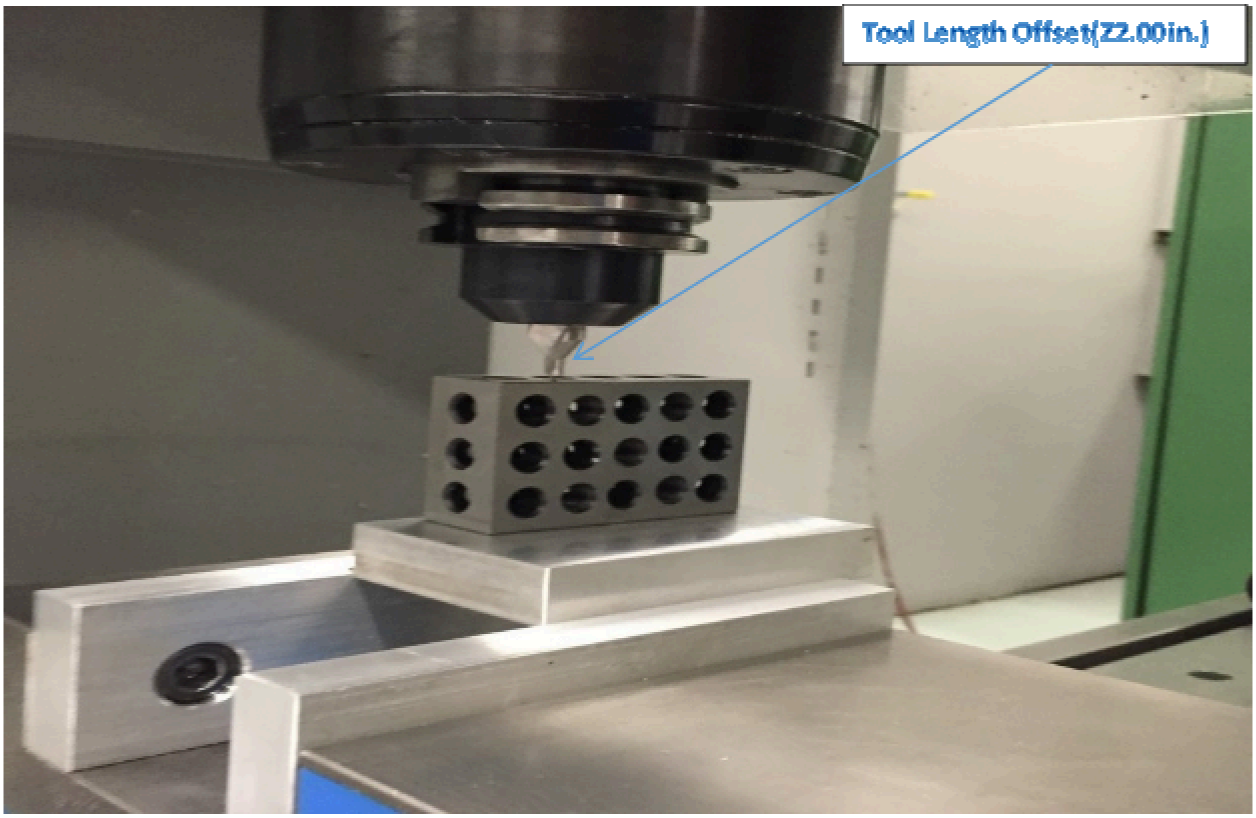


Figure 16. Tool length Offset (2.00 inch above part using 1 2 3 block to verify)

Load CNC Program

1. Edit: Select and Press Edit

2. F1: Select and Press F1

3. Cursor: Press the left arrow key to I/O and then the DN arrow key to move the highlight bar to Disk Directory.

4. Write/Enter: Select and Press Write/Enter

5. Cursor (Disk Directory): Press the DN arrow key to program to load.

6. Write/Enter: Select and Press Write/Enter

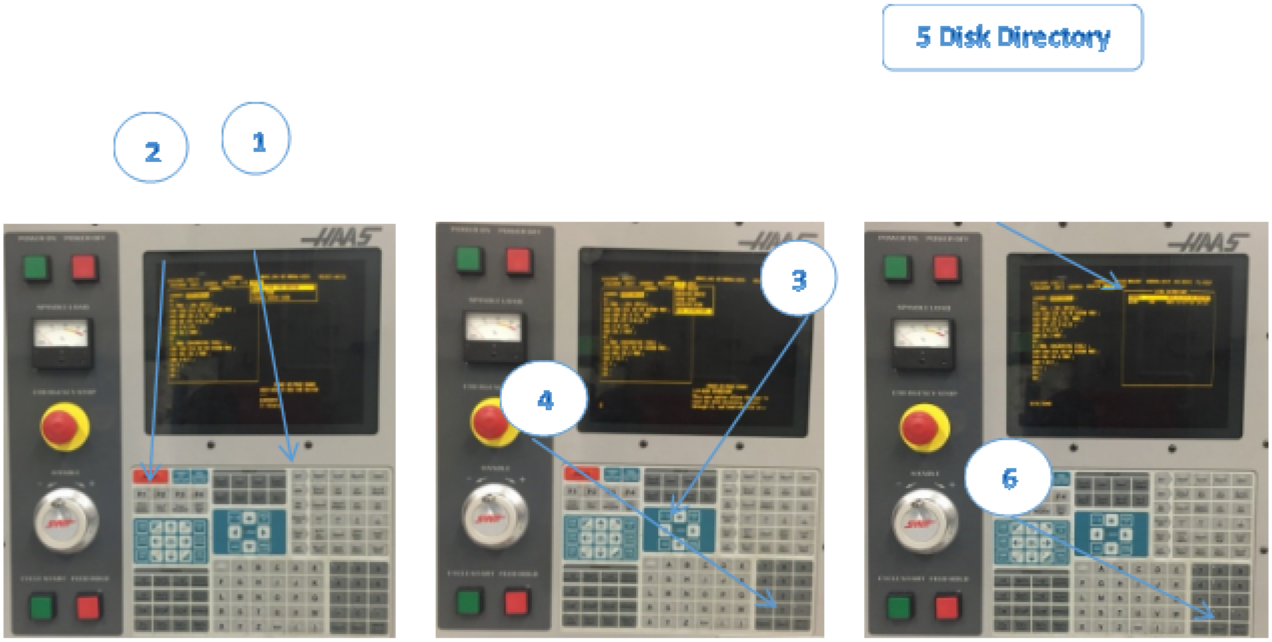


Figure 17. Load CNC Program

Save CNC Program

1. Edit: Select and Press Edit

2. F1: Select and Press F1

3. Cursor: Press the left arrow key to I/O and then the DN arrow key to move the highlight bar to Send Disk.

4. Write/Enter: Select and Press Write/Enter

5. Enter Disk File Name: O80001

6. Write/Enter

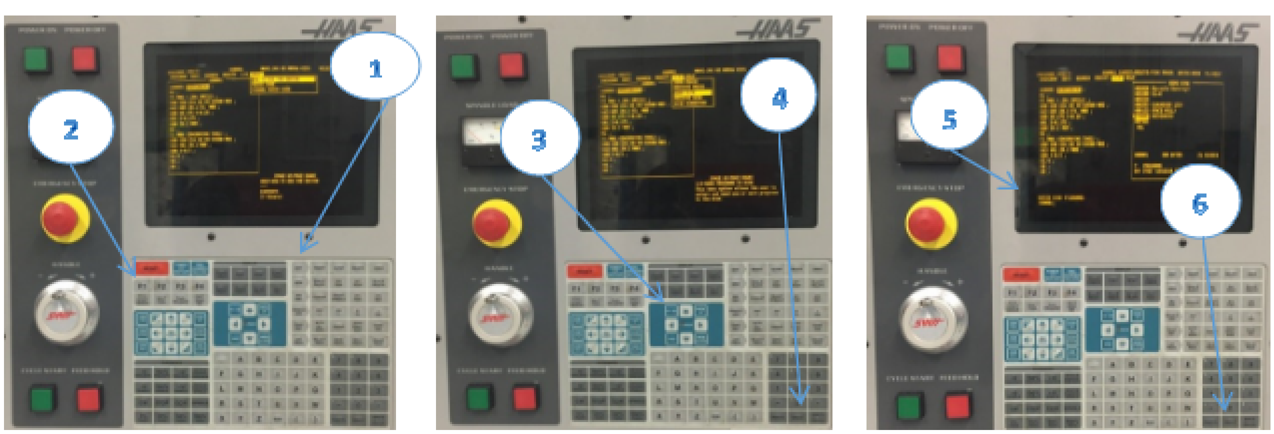


Figure 18. Save CNC Program

Run CNC Program

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

Dry Run Operation

The machine executes all motions exactly as programmed. Do not use a work piece in the machine while dry run is operating. The Dry Run function is used to check a program quickly without actually cutting parts.

To select Dry Run:

1. While in MEM or MDI mode, press (DRY RUN).

When in Dry Run, all rapids and feeds are run at the speed selected with the jog speed buttons.

2. Dry Run can only be turned on or off when a program has finished or [RESET] is

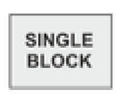
pressed. Dry Run makes all of the commanded X Y Z moves and requested tool

changes. The override keys can be used to adjust the Spindle speeds.

Checklist:

1. Pre-Start

- Ensure vise or fixture is secure and that you have a safe setup.
- There should be no possibility that the work holding will fail to perform as required.
- Remove vise handles.
- Clear the work area of any tools or other objects.
- Close the machine doors.



- Turn Single Block Mode On.
- Press Rapid Feedrate -10 button eight times to set rapid Feed Rate Override to 20% of maximum.

2. Start

- Place one hand on Feed Hold button and be ready to press it in case there are any problems.
- Press Cycle Start Button.

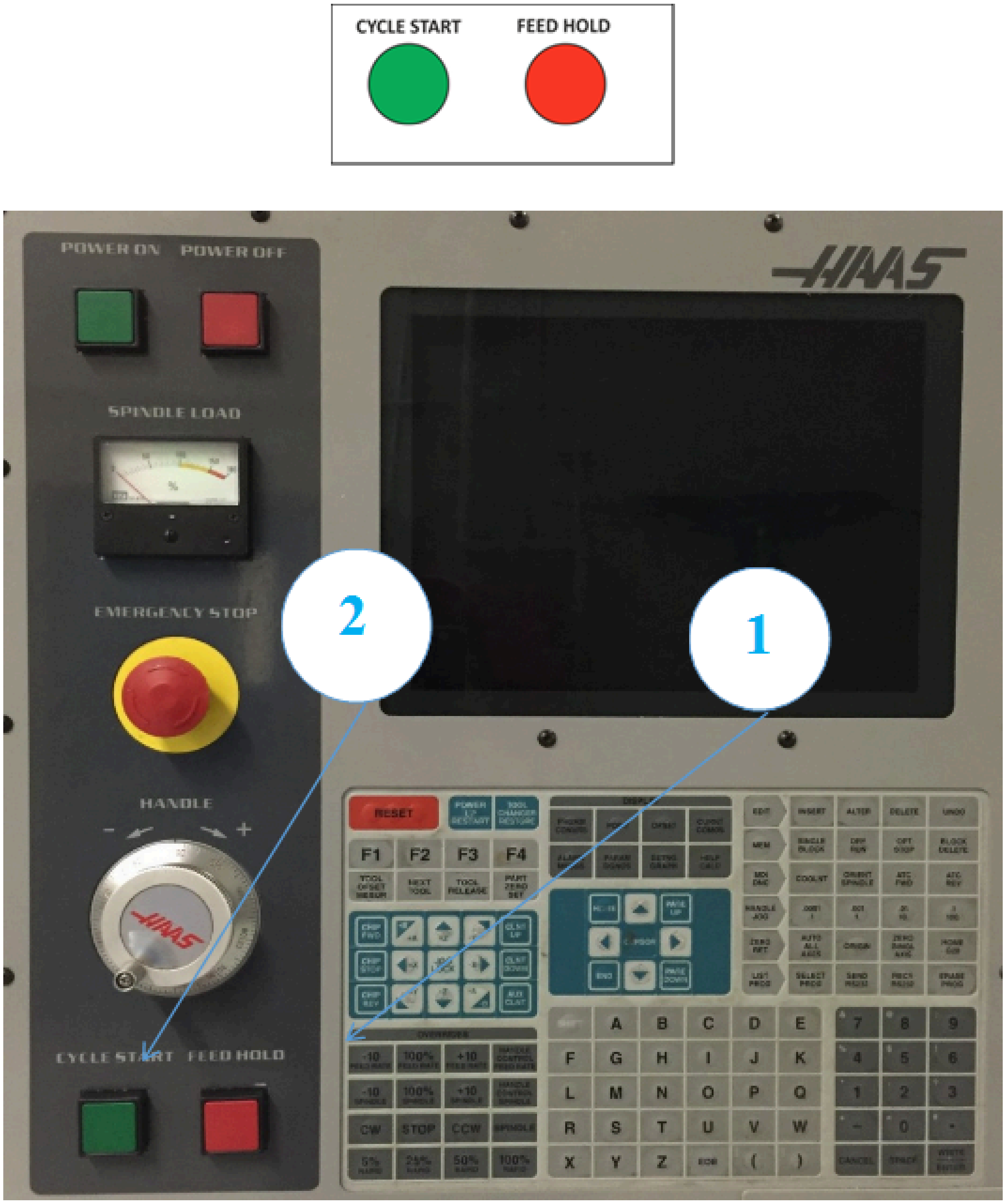


Figure 19. Run CNC Program

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

Adjusting CDC Offsets

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

(Target Feature Size: 2.5000) – (Actual Feature Size: 2.5150) = Wear Value: -0.0150

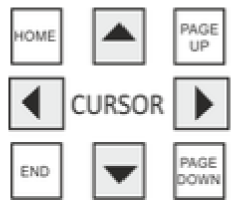
The tool path will now be compensatedfor the size difference. Running the same operation again should result in the feature being exactly the target size

Wear compensation is used only on contour passes. It is not used for face milling, 3D milling, or drill cycles. Select the Wear Compensation option in your CAD/CAM software and, if needed, set a Tool Diameter Wear value as shown above. When used, the wear value is always a negative number. Always set Tool Diameter Geometry to zero for all tools since CAD/CAM software already accounts for the tool diameter by programming the tool center line path.

Checklist:

1. Offset Page: Select and Press Offset

2. Adjust Diameter Offset: Select and Enter Value



- Pg Up/Dn to highlight the tool to be adjusted.
- Enter a value using the numeric keypad.
- Example Tool Diameter Wear value as shown above: -0.0150
- Select and Press Writ/Enter

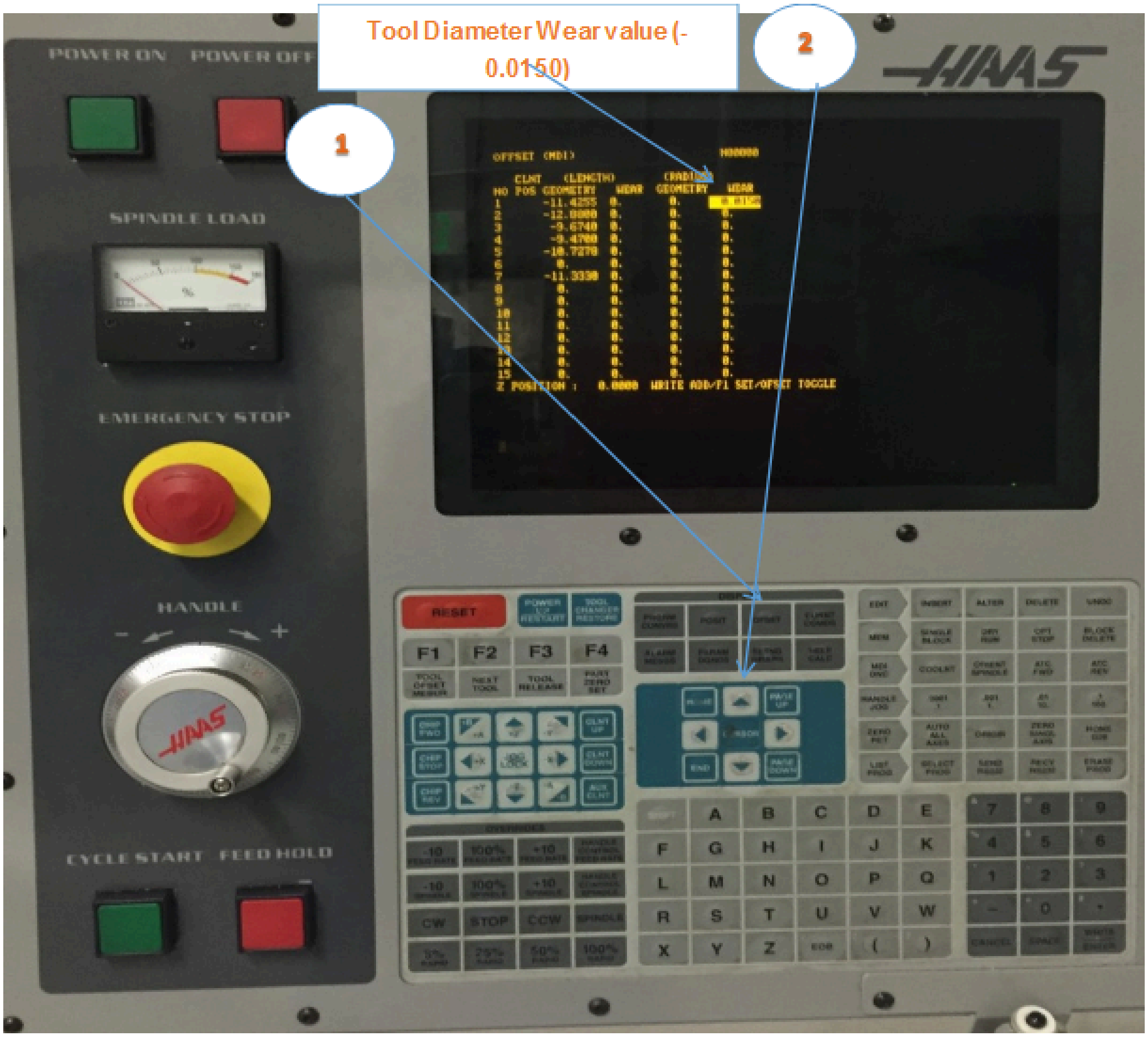


Figure 20. Adjusting CDC Offsets

Shut Down CNCMachine

Checklist:

1.Remove tool from spindle:

- Enter the number of an empty tool carousel.
- Select ATC FWD

2. Jog Machine to Safe Area:

- Select Jog

3.Shut Down Button: Press POWER OFF

Post Power-Down Checklist:

• Wipe spindle with a soft clean rag to remove coolant and prevent rusting.

• Put away tools.

• Clean up work area.

• Always leave the machine, tools, and equipment in the same or better condition than when you found them.

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

UNIT TEST

1. Please lists eight functional areas of HAAS Keyboard.

2. Describe how to Door override.

3. Explain how to load Tools.

4. Describe MDI and give one example.

5. Explain how to set part zero offset.

6. Please describe using MDI to verify part zero offset.

7. Explain how to set Tool Length offset

8. Please describe using MDI to verify Tool Length offset.

9. Describe how to save CNC program.

10. Please Explain the shut down procedure.

24

Unit 7: Mastercam

OBJECTIVE

After completing this unit, you should be able to:

- Describe save image.
- Describe the load image to Mastercam.
- Describe scaling and dragging or translate image.
- Describe Mastercam setup, stock setup, and Tool setting.
- Describe creating Toolpaths.

Mastercam

Raster To Vector Images Conversion:

This Worksheet walks you through how to import Raster images into Mastercam. Mastercam is a vector software, where most images for the internet and other sources are Raster Images. Raster images are made up of thousand of pixels of differing color, where vector images are image of line that use mathematical formulas to determine their shape. In this activity you will learn how to convert Raster images to Vector images that Mastercam can use.

1. Using the internet search for your desired Image. Logos or images with sharp color changer work best, try to avoid pictures taken with a camera. .jpg, .gif, .bmp are the file extension currently supported by Mastercam.

2. Try to get the larges image possible. When Using Google examine the images sizing information located under the image. The Larger the number the better.

3. Then navigate to the raw picture by clicking on the image.

4. Click on See full size image.

5. RIGHT CLICK on the image and select Save Image As…

6. Save your raster image to a location that you know.

7. Launch Mastercam 2017

8. Once it loads Press ALT-C or Run User Application (under Setting) a window will open called Chooks. This is a collection of add on file to mastercam.

9. Navigate and click on Rast2Vec.dli

10. It many ask you of you want to keep current Geometry. Then, Click Yes

11.Navigate to your saved image and click on it.

12. The Black White conversion window will open. Slide the Threshold slider in the Black White conversion dialog box until the image on right shows the desired amount of detail.

13. When you like what you see. Then, click OK.

14. Rast2Vec window will popup no modification should be needed. Then, click OK.

15. When the Adjust Geometry opens. Then, click OK.

16. Your image should now be on the screen.

17. Click Yes to exit Rast2Vec.

Scaling and Dragging or Translate your image

After you have imported a image it often not in the right location or the proper size that you need. You need to Scale it up or down and Drag it to the position that you would like.

Scaling:

1. From The ToolBar >Xform>Scale

2. Then Select all the line that part of your image. The Line you select should turn yellow. You can use a window or select each line individually. If you pick a line you do not what just pick it again and it should change back form yellow.

3. Once you have selected your entire image then click on the Green Ball.

4. The Scale window will open. Change To MOVE and to PERCENTAGE, and then adjust the percent up or down until your image is the size you would like. After typing the new percentage number hit ENTER on the keyboard. This will preview that size change. Once the size is what you are looking for. Then, click OK.

Dragging or Translate

1. From the Tool Bar>xfrom>Drag or Translate

2. Once again select all the line that part of your image.

3. Once you have selected your entire image then click on the Green Ball.

4. Change from Copy To Move.

5. Click near your Image in the graphics screen. As you move your mouse the image will move.

6. Once the image is in the proper place LEFT Click again and it will place the image.

MASTERCAM SETUP

1. Turn on the Operation Manager window by pressing Alt-O

2. Click on Machine Type – Mill, then select the HASS 3X MINI MILL – TOOLROM.MMD-5. You should see the HASS 3X MINI pop up in the Operation Manager window.

STOCK SETUP

1. Click on the plus sign next to Properties to see the drop down list and click on stock setup. The Machine Group Properties dialog box should appear. Check the box next to Display to activate it, then click on the Bounding Box button. The Bounding Box dialog box should appear. Confirm that X, Y AND Z are all be set to zero then click OK.

2. Change the value of Z to stock you are using, then click OK. On an isometric view, you should see your part go from 2D to 3D with the image on the top surface.

TOOL SETTING

1. Click on Tool Setting. Enter a program number….Feed Calculation should be "From Tool." Under Toolpath Configuration, check "Assign tool number sequentially" and "Warn of duplicate tool numbers." Under Advanced Operation, check the "Override defaults with modal values" box and then check all three selections below it. Under Sequence #, change the start number to 10. Select the material by clicking on the Select button. Click on the drop down arrow associated with Source and select "Mill – library." From the list, select "ALUMINUM inch – 6061" then click OK. Click OK to exit the Machine Group Properties dialog box.

CREATING TOOLPATHS

1. Choose Toolpath – Contour. The chaining dialog box should appear. Select Window to choose your engraving elements, then click anywhere to establish an approximate start point (selection will change to yellow). Click OK to bring up the 2D Toolpaths – Contour dialog box.

2. Toolpath Type should automatically be set to Contour.

3. Click on Tool (below Toolpath Type) then click on Select Library Tool button. A Tool Selection dialog box will appear. To limit the list to a particular type of tool (ball end mills which will use for engraving), Click on Filter then select/de-select tool types so that only Endmill2 Sphere is highlighted. Click OK. Choose the 1/32 Ball Endmill followed by click OK. Change federate to 5.0 and Spindle Speed to 4000 then click OK.

4. Under Cut Parameters, Compensation Type should be off.

5. Under Lead In/Lead Out, Uncheck the Lead In/Out box as will not be using this feature.

6. Under Linking Parameters, Clearance should be set to 0.5, Retract should be 0.1, feed Plane should be 0.1, Top of stock should be O.0 and Depth should be -0.015. Then click OK.

7. Run Verify Selected Operations in the Operations Manager to see the toolpath.

UNIT TEST

1. What software is Mastercam is?

2. Please explain the Threshold Slider.

3. Lists file supported by Mastercam.

4. Describe scaling and dragging or translate image.

5. Describe stock setup.

6. Describe the tool setting.

7. Explain how to create Toolpaths.