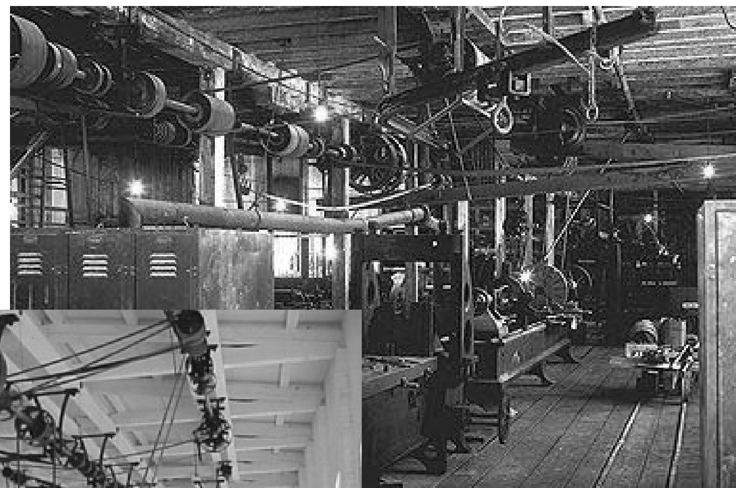
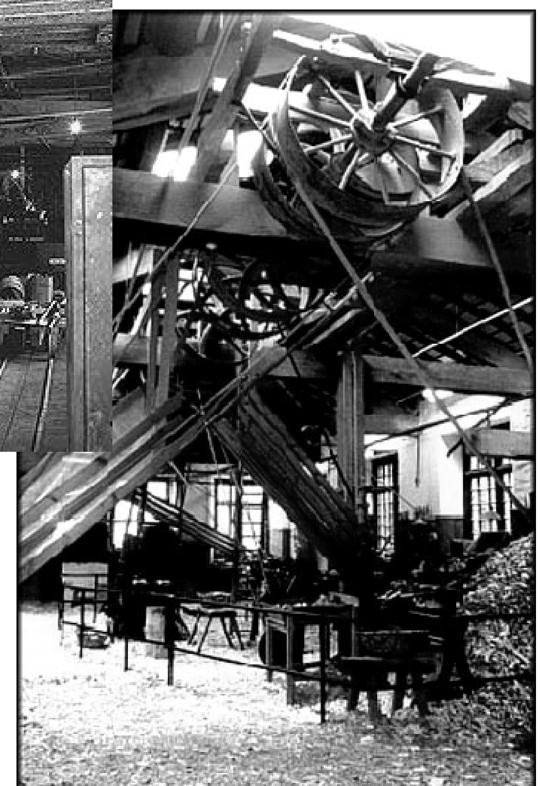
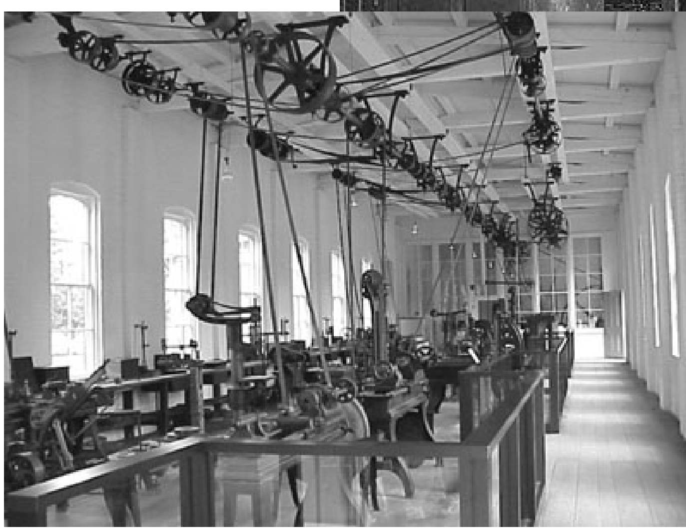
CNC Applications

History and Terminology







Background & Definitions (Chapter 1)

- Requirements for a skilled machinist
  - Serve a 4 year apprenticeship including classes in algebra, trigonometry, print reading, and drafting along with 8,000 hours of on-the-job-training.
  - The machinist must purchase several thousand dollars worth of precision tools.
  - Machinists often make a lower hourly wage than other skilled trades such as electricians and plumbers.
  - Production operations often require a very skilled person to perform the same operations over and over which most machinists find boring.

Background & Definitions (continued)

- During the 1930’s and 1940’s, there was much labor unrest between machinists and management at large companies. Work stoppages and strikes angered management.
- At the same time, World War II increased the complexity of parts required for common products.
- The most complicated product at the time was the jet aircraft which required large quantities of complex, high-precision components.

Background & Definitions (continued)

- The combination of labor problems and more complicated components precipitated the introduction of automatic machines that could be programmed to produce different parts.

- Automatic machines had been available since the US Civil War (1861-1865), but the machines could only produce one part and required large amounts of time to set up to produce a different part.

- An electronically controlled machine that could be easily changed to produce a different part was required.

Background & Definitions (continued)

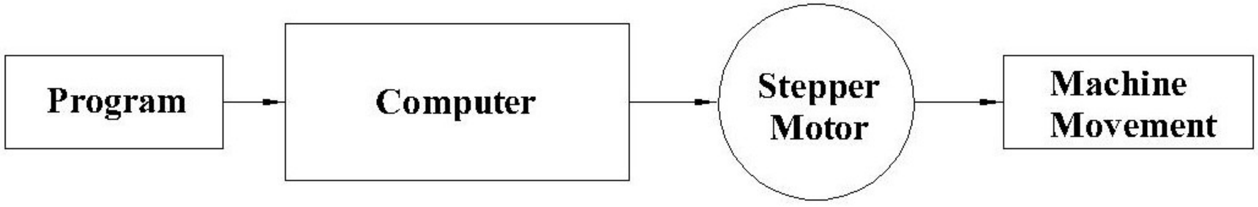
- NC – Numerical Control
  - The first successful electronically programmed automatic machine was a joint project between Massachusetts Institute of Technology (MIT) and the US Air Force in the mid 1950's. It was a three axis milling machine controlled by a room full of vacuum tube electronics. Even though it was unreliable, it set the stage for modern machines. The controller was called Numerical Control, or NC.
  - The Electronics Industry Association (EIA) defines NC as "a system in which actions are controlled by the direct insertion of numerical data at some point.“
  - NC machines were controlled electronically, without the use of a computer.

Background & Definitions (continued)

- CNC – Computer Numerical Control
  - CNC machines use a computer to assist and improve functionality of number and code control.
  - In the 1960's, CNC machines became available with timesharing on mainframe computers. True NC machines continued to be built.
  - By the 1970's, specialized computers were being manufactured for CNC controls. By the late 1970's, no true NC machines were being made, only CNC.
  - During the 1980's, many machine manufactures took advantage of PC technology to increase the reliability and decrease the cost of CNC controls.
  - Today, all machines are CNC although the term NC is still used, but not in its original definition.

Machine Control Systems

- Stepper Motor Control



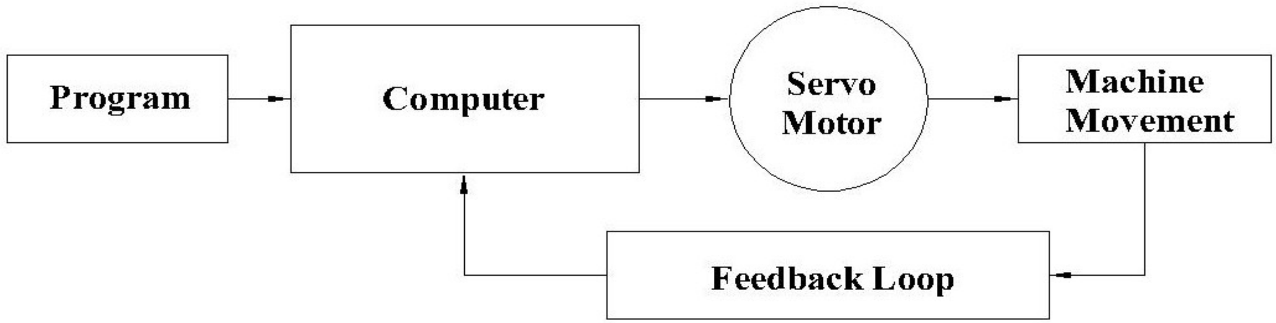
➢The stepper motor takes voltage pulses and converts them to rotary motion. If the machine resolution (smallest motion) is 0.0001” and you want to move 3”, the computer sends 30,000 (30,000x0.0001”=3.0”) pulses to the motor and the machine moves 3”.

➢Problem: stepper motors have limited torque, and if excess pressure is applied, the motor will slip and the machine loses its position. Then, the operator must restart the machine.

➢The machine does not know where it actually is, only where it should have moved. This method works fine unless the motor slips.

Machine Control Systems (continued)

- Servo Motor Control



➢The servo motor has a feedback loop to check the machine's actual position. If the program tells the computer to move 3", the servo motor starts turning and does not stop until the feedback loop tells the computer that the machine has actually moved 3"

➢Advantage: servo motors have high torque capabilities to take heavy cuts at high speeds. It stops and gives an alarm when the motor is over-torqued.

➢Advantage: the machine always knows its actual position.

Modern CNC Machine Characteristics

- Massive, usually four times heavier than an equivalent conventional (manual) machine.
- Large motors with high speed capabilities to take advantage of modern cutting tools. Horsepower and spindle speeds are generally four to ten times faster than conventional machines.
- Automatic tool changers that hold from eight to hundreds of cutting tools that are quickly changed under program control.
- High accuracies. The minimum resolution of most machines is 0.0001” or 0.001mm, and some machines are capable of manufacturing parts to that accuracy, depending on the process. Ball screws practically eliminate backlash (slop) in the movement screws.

Modern CNC Machine Accuracy

- Accuracy of CNC machines depends on their rigid construction, care in manufacturing, and the use of ball screws to almost eliminate slop in the screws used to move portions of the machine. These pictures show the precision balls which re-circulate in the nut.

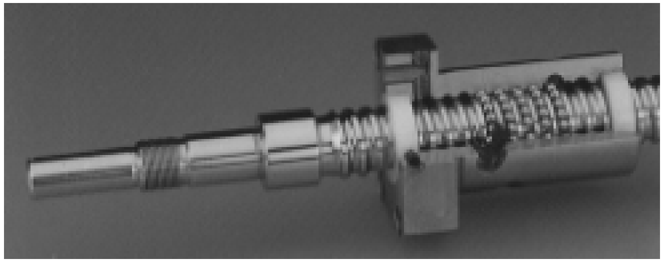
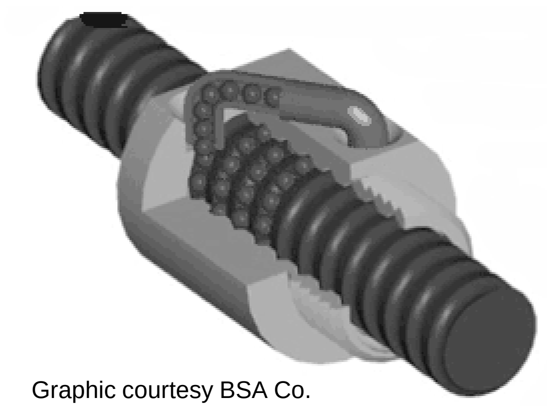


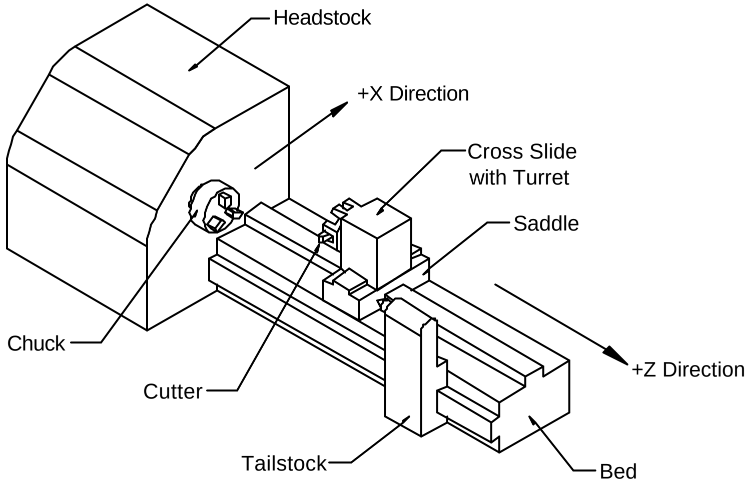
Photo courtesy Thompson Ball Screw.



Graphic courtesy BSA Co.

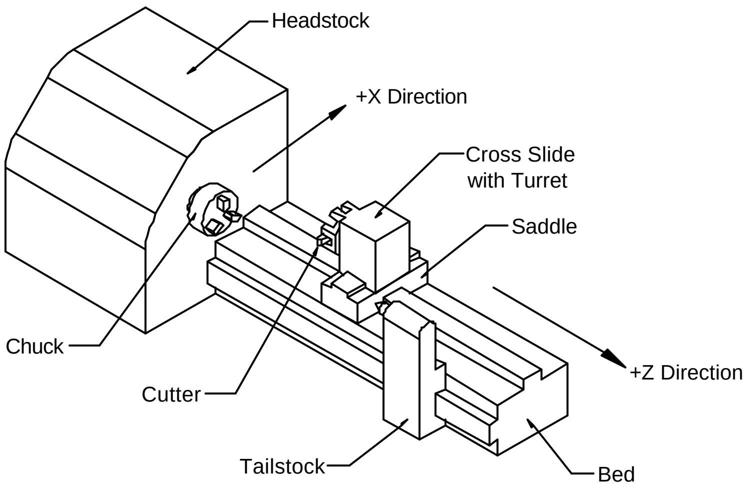
CNC Applications

Introduction to Turning Centers



Turning Centers

A Turning Center is simply a CNC lathe with a multi-station turret and an enclosure.



Notice the turret attached to the cross slide.

Note: the CNC control and the enclosure are not shown.

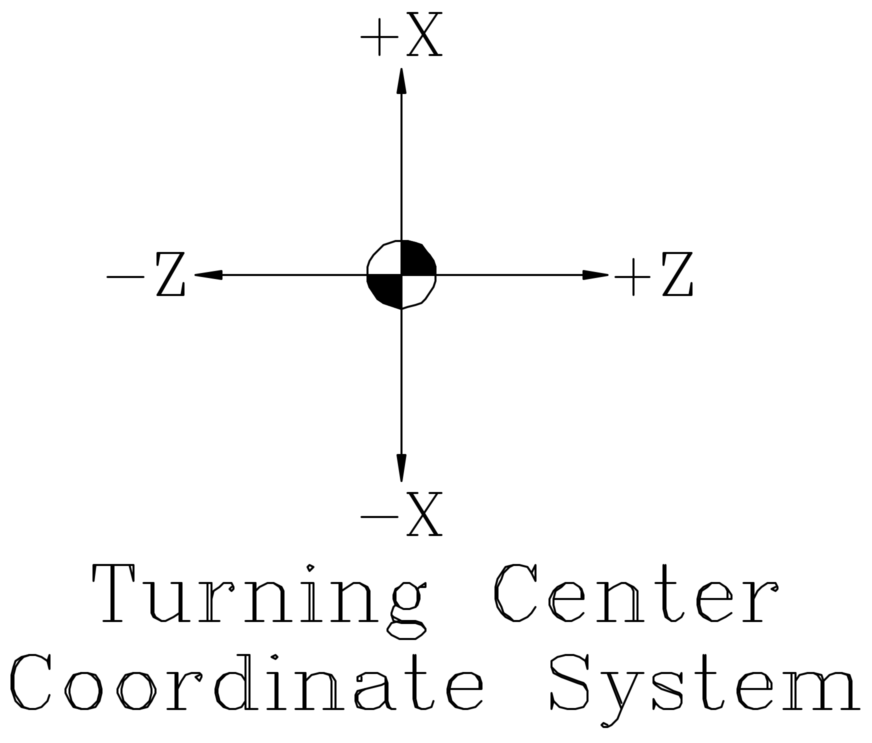
Characteristics of Turning Centers

- Turret is on the far side to ease part loading and unloading.
- Heavy for increased rigidity.
- High spindle speeds to effectively use hard cutters.
- Powerful motors.
- Communication capabilities.

Haas SL-20
A Typical Turning Center

The Haas SL-20 turning center has a 20 HP motor, 45-4000 RPM spindle, ten tools in the turret, 8" cutting diameter and 20" between centers. It weighs 9000 lbs.

Turning Center Coordinate System



Z is the length of the part.

X is the diameter of the part.

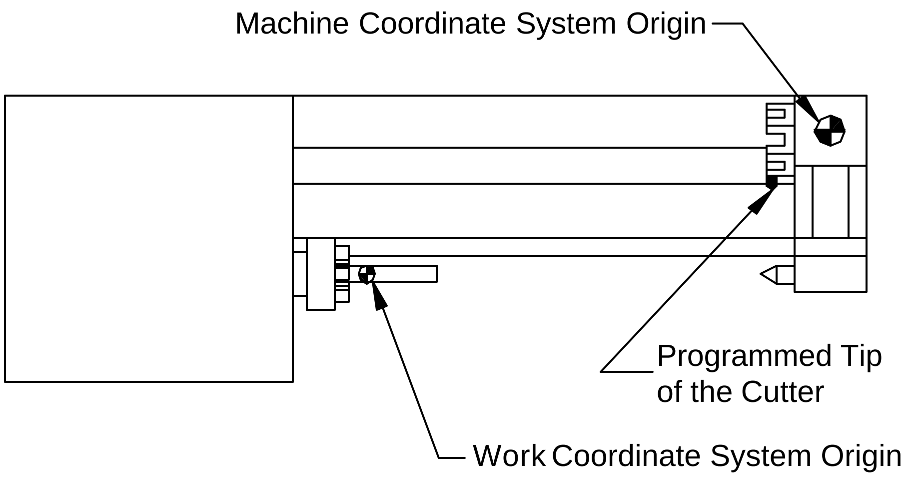
The partially shaded circle represents the origin.

The programmer chooses the location of the origin on the part, usually the back, center or front, center.

We will always use the back, center since it simplifies machine set-up.

Program to move the tool, not the work.

The Coordinate System Problem



Top View of the Turning Center

The machine operates from the Machine Coordinate System (MCS). Note that all MCS movements are in the negative X and Z directions.

We program from the Work Coordinate System (WCS).

The tip of the cutter is offset from the MCS and is different for each cutter.

The Solution

- We can easily measure the distance from the MCS to the WCS and enter it into the machine control. This is called a **Fixture Offset**.
- We can easily measure the distance from the MCS to the tip of the each cutter and enter the values in the machine control. These values are called **Tool Offsets**.

Separating the Programmer from the Machine

- The programmer chooses the WCS on the centerline of the part, generally at the back, and then programs the cutter movement from there.
- The programmer calls the correct fixture offset and tool offset numbers in the program (we'll cover these codes later).
- So, the programmer does not have to be concerned with any machine specific measurements.

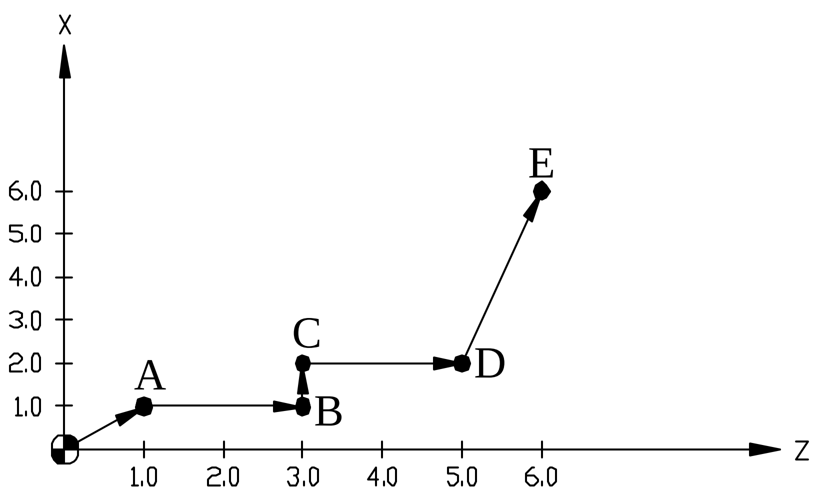
Incremental vs. Absolute Programming

Suppose we want to move from the origin through points A, B, C, D, and E. We can move:

- incrementally – distance from previous point,

or

- absolutely – new distance from the origin.



	Incremental		Absolute	
Point	Z	X	Z	X
Origin	0	0	0	0
A	1	1	1	1
B	2	0	3	1
C	0	1	3	2
D	2	0	5	2
E	1	4	6	6

Incremental vs. Absolute Programming (continued)

- Notice in the previous table that each move in incremental mode is the distance from the previous point, while each move in absolute is the distance from the origin, regardless of the previous point.
- Most programmers initially think incremental programming is easier.
- However, editing for program changes is much easier in absolute mode.
- About 95% of all programming is done in absolute mode.
- The remaining 5% is for special cases such as repetitive features where incremental can be a real time saver.

CNC Applications

Speed and Feed Calculations

Photo courtesy ISCAR Metals.

Turning Center Cutters

What types of cutters are used on CNC turning Centers?

- Carbide (and other hard materials) insert turning and boring tools
- High Speed Steel (HSS) drills and taps

Where do I find information for calculating RPM and feed rates?

- Cutting tool manufacturer (first choice)
- *Machining Data Handbook*
- *Machinery's Handbook* (we'll use this option)

Standard Insert Shapes

V – used for profiling, weakest insert, 2 edges per side.

D – somewhat stronger, used for profiling when the angle allows it, 2 edges per side.

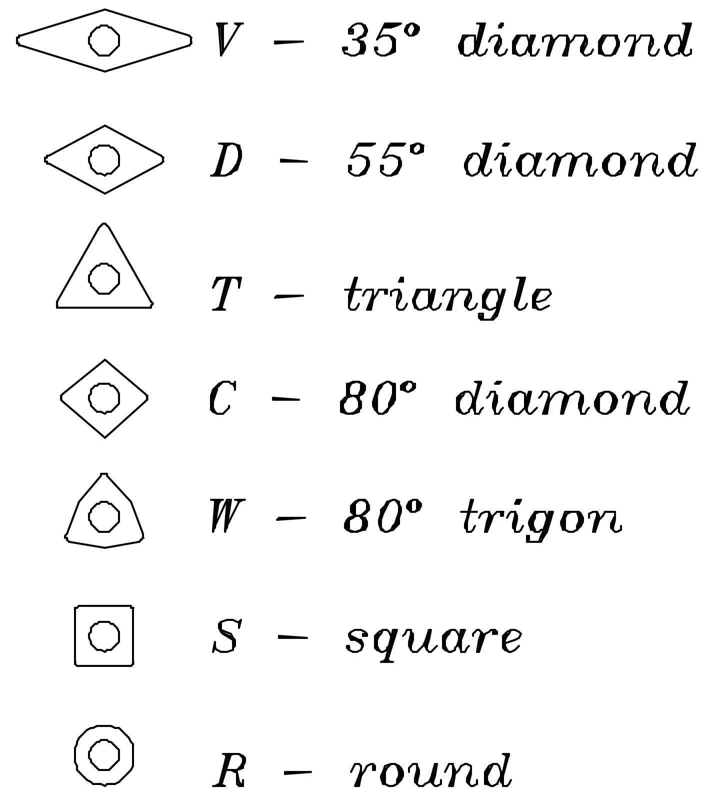
T – commonly used for turning because it has 3 edges per side.

C – popular insert because the same holder can be used for turning and facing. 2 edges per side.

W – newest shape. Can turn and face like the C, but 3 edges per side.

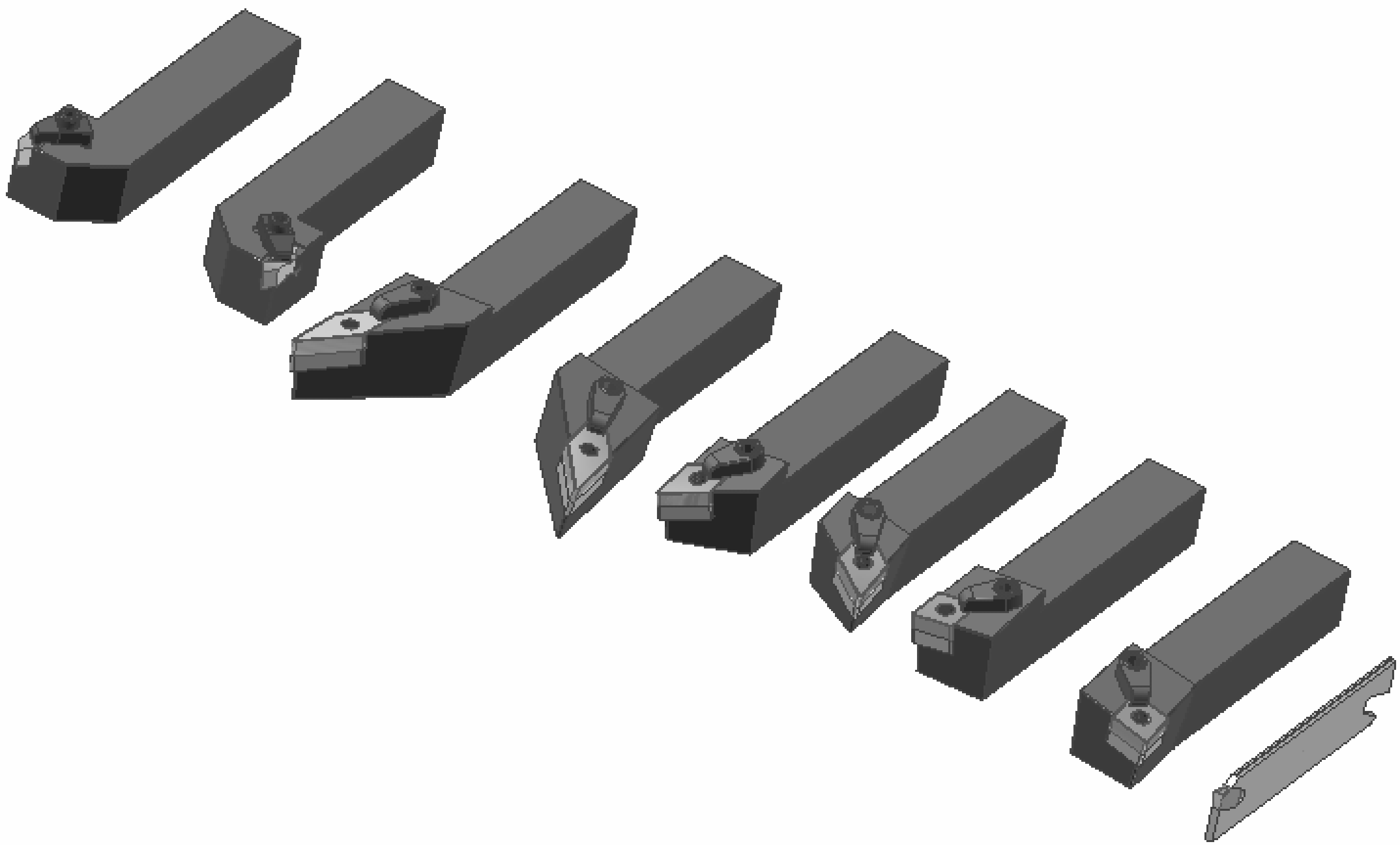
S – Very strong, but mostly used for chamfering because it won't cut a square shoulder. 4 edges per side.

R – strongest insert but least commonly used.



See the "Tooling" thumb tab in the *Machinery's Handbook*.

Typical Turning, Threading, and Parting Tools



Tool Holder Hand

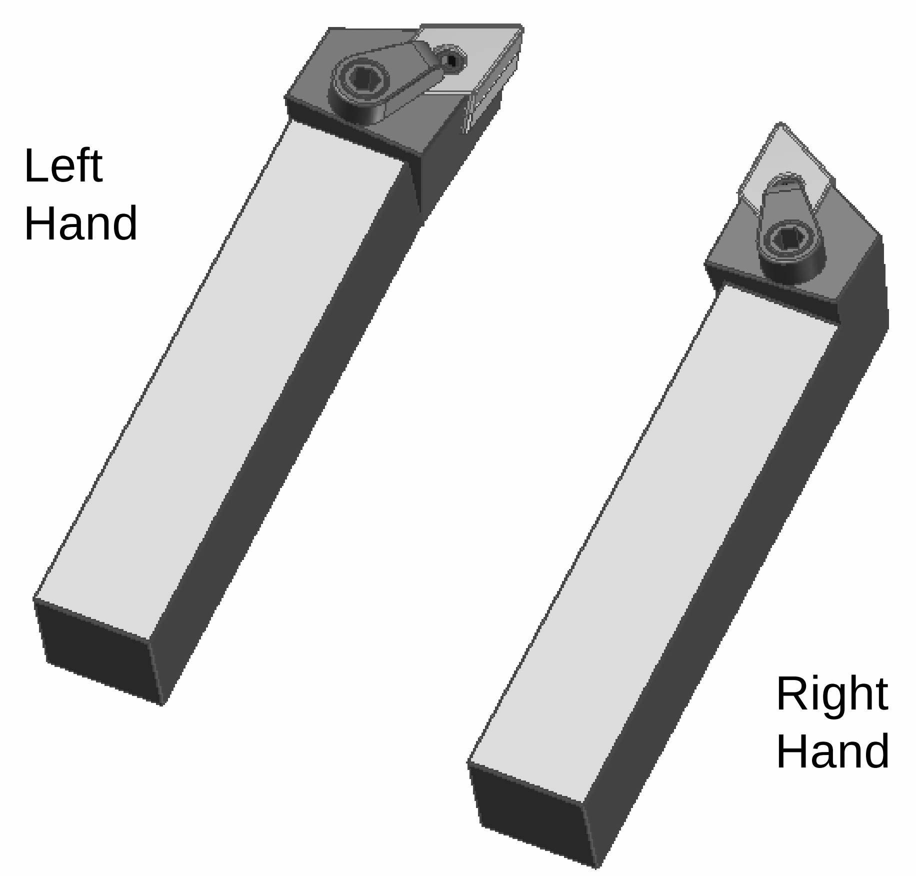
For most CNC turning centers, the cutter is on the back side of the part and is upside down.

Right Hand tool then turns towards the chuck.

Left Hand tool then turns towards the tailstock.

If the cutter is symmetrical with the shank, it is called NEUTRAL HAND.

See the “Tooling” thumb tab in the *Machinery’s handbook* for more information.



Single Point Indexable Insert Holders

Tooling Considerations

- Tooling choices depend on the type of workpiece, the machine, and the desired surface finish.
- Harder workpieces require harder cutters.
- Modern cutters require the turning center to have high spindle speeds and powerful motors.

Cutting Speed

- What is cutting speed?
  - Not RPM
  - Relative speed of the work and cutter
  - Units in feet/minute (fpm)
  - Usually designated as V, cs, or S
  - Tabulated in the book based on material, cutter type, and type of cut (roughing or finishing)
  - Needed to calculate RPM

Calculating Turning RPM

- The formula for calculating RPM is given on page 1016 as:

$$N = RPM = \frac{12 \times V}{p \times D}$$

Where V = cutting speed to be looked up in the handbook

p = 3.14

D = diameter being cut

When punching buttons on your calculator, do this:

$$12 \times V \div p \div D =$$

Note the difference between this and the actual formula.

To use this formula, we must first find V in the handbook

See page 1022 for a list of the tables.

Types of Cuts

- Roughing – primary considerations:
  - Just removing metal, surface finish does not matter.
  - Requires a strong cutter.
  - Generally have deep depth of cuts and fast feed rates.
  - The cutting speed is generally adjusted slower to keep heat down.

Types of Cuts (continued)

- Finishing – primary considerations:
  - Must meet required surface finish and size specifications.
  - Requires a hard cutter to hold its shape well.
  - Generally have small depth of cuts and slow feed rates.
  - The cutting speed is generally adjusted upward to give a better surface finish.

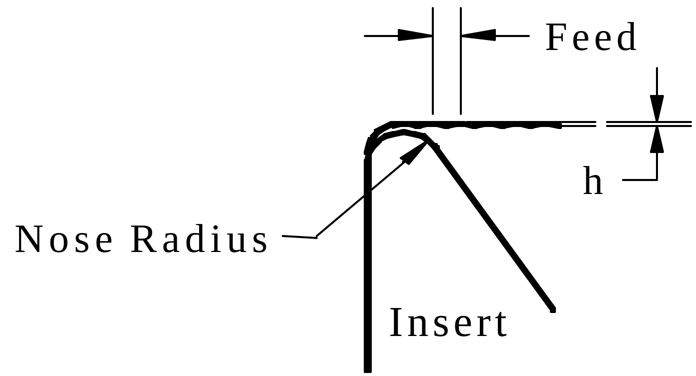
Surface Finish Requirements

The surface finish depends on the feed rate and on the cutter nose radius.

Generally, a large nose radius and a slow feed rate coupled with high cutting speed gives the best finish.

However, too large of a nose radius induces chatter ruining the finish and the size.

Most inserts use a 1/32” nose radius as a good compromise.



$$h = \frac{f^2}{8 \times r}$$

h = height of marks
f = feed rate
r = nose radius

and

$$roughness = \frac{f^2}{32 \times r}$$

Actual roughness in inches, usually report in microinches.

General Feed and Depth of Cut Recommendations

- Roughing:
  - 0.1” to 0.25” depth of cut (radial)
  - 0.012 inches per revolution (ipr) to 0.018 ipr feed rate
- Finishing:
  - 0.03” to 0.05” depth of cut (radial)
  - 0.006 ipr to 0.010 ipr feed rate

  Note 1: the depth of cut should not be less than the tool nose radius unless special finishing inserts are being used.

  Note 2: smaller feed rates can be used if special finishing inserts are being used.

Calculating RPM for Turning Operations with Hard Cutters

- Use this procedure for carbide, ceramic, and cermet inserts.
- We will adjust the cutting speed based on the desired depth of cut and feed rate.

Calculating Turning RPM (continued)

Six step process:

1. Select depth of cut - as deep as possible.
2. Select feed - appropriate for roughing or finishing.
3. Find the original cutting speed in the tables. (See the listing on page 1022 for the appropriate table.)
4. Find the feed and depth of cut factors in Table 5a, page 1035.
5. Modify the original cutting speed based on step 4.
6. Calculate the RPM.

Note: All data will be found in the “Machining” thumb tab in the *Machinery’s Handbook*.

Calculating Turning RPM (continued)

Example:

Take 0.250 depth of cut, 0.012 feed in quenched and tempered 8620 steel with a Brinell hardness of 300, hard coated carbide cutter, 2.5” diameter part.

Step 1:

Depth of cut given at 0.25”.

Step 2:

Feed rate given as 0.012 ipr.

Calculating Turning RPM (continued)

Step 3: From Table 1, page 1029, locate cutting parameters for this material

Table 1. *(Continued)* **Cutting Feeds and Speeds for Turning Plain Carbon and Alloy Steels**

Material AISI/SAE Designation	Brinell Hardness	HSS Speed (fpm)		Uncoated Carbide Hard Opt.	Uncoated Carbide Hard Avg.	Uncoated Carbide Tough Opt.	Uncoated Carbide Tough Avg.	Coated Carbide Hard Opt.	Coated Carbide Hard Avg.	Coated Carbide Tough Opt.	Coated Carbide Tough Avg.	Ceramic Hard Opt.	Ceramic Hard Avg.	Ceramic Tough Opt.	Ceramic Tough Avg.	Cermet Opt.	Cermet Avg.
				Tool Material: f = feed (0.001 in./rev), s = speed (ft/min)													
Free-machining alloy steels: (leaded): 41L30, 41L40, 41L47, 41L50, 43L47, 51L32, 52L100, 86L20, 86L40	150–200	120	f	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	730	990	300	430	1090	1410	780	1105	1610	2780	1345	2005	1355	1695
	200–250	100	f	17	8	36	17	17	8	28	13	13	7	13	7	7	3
			s	615	815	300	405	865	960	755	960	1400	1965	1170	1640	1355	1695
	250–300	75															
	300–375	55	f	17	8	36	17	17	8	28	13	10	5	10	5		
			s	515	685	235	340	720	805	650	810	1430	1745	1070	1305		
	375–425	50															
Alloy steels: 4012, 4023, 4024, 4028, 4118, 4320, 4419, 4422, 4427, 4615, 4620, 4621, 4626, 4718, 4720, 4815, 4817, 4820, 5015, 5117, 5120, 6118, 8115, 8615, 8617, 8620, 8622, 8625, 8627, 8720, 8822, 94B17	125–175	100															
	175–225	90	f	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	525	705	235	320	505	525	685	960	1490	2220	1190	1780	1040	1310
	225–275	70	f	17	8	36	1	17	8	28	13	10	5	10	5	7	3
			s	355	445	140	200	630	850	455	650	1230	1510	990	1210	715	915
	275–325	60	f	17	8	36	17	17	8	28	13	9	5	8	5	7	3
			s	330	440	135	190	585	790	240	350	1230	1430	990	1150	655	840
	325–35	50															
	375–425	30 (20)	f	17	8	36	17	17	8	28	13	8	4	8	4	7	3
			s	330	440	125	175	585	790	125	220	1200	1320	960	1060	575	740
Alloy steels: 1330, 1335, 1340, 1345, 4032, 4037, 4042, 4047, 4130, 4135, 4137, 4140, 4142, 4145, 4147, 4150, 4161, 4337, 4340, 50B44, 50B46, 50B50, 50B60, 5130, 5132, 5140, 5145, 5147, 5150, 5160, 51B60, 6150, 81B45, 8630, 8635, 8637, 8640, 8642, 8645, 8650, 8655, 8660, 8740, 9254, 9255, 9260, 9262, 94B30 E51100, E52100 use (HSS Speeds)	175–225	85 (70)	f	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	525	705	235	320	505	525	685	960	1490	2220	1190	1780	1020	1310
	225–275	70 (65)	f	17	8	36	17	17	8	28	13	10	5	10	5	7	3
			s	355	445	140	200	630	850	455	650	1230	1510	990	1210	715	915
	275–325	60 (50)	f	17	8	36	17	17	8	28	13	9	5	8	5	7	3
			s	330	440	135	190	585	790	240	350	1230	1430	990	1150	655	840
	325–375	40 (30)															
	375–425	30 (20)	f	17	8	36	17	17	8	28	13	8	4	8	4	7	3
			s	330	440	125	175	585	790	125	220	1200	1320	960	1060	575	740

Material

Hardness

Cutting Parameters

Calculating Turning RPM (continued)

Step 3 (continued):

From Table 1, page 1029, we find

V_{opt} = 585 fpm ipr V_{avg} = 790 fpm F_{opt} = 0.017

Note that the table lists cutting speed as S rather than V as used everywhere else. Note that the feed rates are given in 0.001 ipr, so the 17 listed for F_{opt} is actually 0.017 ipr.

Calculating Turning RPM (continued)

Step 4:

Once we have located the optimum and average cutting speeds and the optimum feed, we finish our calculation using the data and process described in Table 5A, page 1035.

Calculating Turning RPM (continued)

First, calculate the ratio of the feeds.

Second, calculate the ratio of the cutting speeds

Fourth, determine the nearest depth of cut and lead angle column

Fifth, find the depth of cut factor where the step 1row crosses the step 4 column.

Table 5a. Turning-Speed Adjustment Factors for Feed, Depth of Cut, and Lead Angle

Ratio of Chosen Feed to *Optimum* Feed	Ratio of the two cutting speeds given in the tables V_{avg}/V_{opt}							Depth of Cut and Lead Angle									
								1 in. (25.4 mm)		0.4 in. (10.2 mm)		0.2 in. (5.1 mm)		0.1 in. (2.5 mm)		0.04 in. (1.0 mm)	
	1.00	1.10	1.25	1.35	1.50	1.75	2.00	15°	45°	15°	45°	15°	45°	15°	45°	15°	45°
	Feed Factor, F_f							Depth of Cut and Lead Angle Factor, F_d									
1.00	1.0	1.0	1.0	1.0	1.0	1.0	1.0	0.74	1.0	0.79	1.03	0.85	1.08	1.0	1.18	1.29	1.35
0.90	1.00	1.02	1.05	1.07	1.09	1.10	1.12	0.75	1.0	0.80	1.03	0.86	1.08	1.0	1.17	1.27	1.34
0.80	1.00	1.03	1.09	1.10	1.15	1.20	1.25	0.77	1.0	0.81	1.03	0.87	1.07	1.0	1.15	1.25	1.31
0.70	1.00	1.05	1.13	1.22	1.22	1.32	1.43	0.77	1.0	0.82	1.03	0.87	1.08	1.0	1.15	1.24	1.30
0.60	1.00	1.08	1.20	1.25	1.35	1.50	1.66	0.78	1.0	0.82	1.03	0.88	1.07	1.0	1.14	1.23	1.29
0.50	1.00	1.10	1.25	1.35	1.50	1.75	2.00	0.78	1.0	0.82	1.03	0.88	1.07	1.0	1.14	1.23	1.28
0.40	1.00	1.09	1.28	1.44	1.66	2.03	2.43	0.78	1.0	0.84	1.03	0.89	1.06	1.0	1.13	1.21	1.26
0.30	1.00	1.06	1.32	1.52	1.85	2.42	3.05	0.81	1.0	0.85	1.02	0.90	1.06	1.0	1.12	1.18	1.23
0.20	1.00	1.00	1.34	1.60	2.07	2.96	4.03	0.84	1.0	0.89	1.02	0.91	1.05	1.0	1.10	1.15	1.19
0.10	1.00	0.80	1.20	1.55	2.24	3.74	5.84	0.88	1.0	0.91	1.01	0.92	1.03	1.0	1.06	1.10	1.12

Use with Tables 1 through 9. Not for HSS tools. Tables 1 through 9 data, except for HSS tools, are based on depth of cut = 0.1 inch, lead angle = 15 degrees, and tool life = 15 minutes. For other depths of cut, lead angles, or feeds, use the two feed/speed pairs from the tables and calculate the ratio of desired (new) feed to *optimum* feed (largest of the two feeds given in the tables), and the ratio of the two cutting speeds (V_{avg}/V_{opt}). Use the value of these ratios to find the feed factor F_f at the intersection of the feed ratio row and the speed ratio column in the left half of the table. The depth-of-cut factor F_d is found in the same row as the feed factor in the right half of the table under the column corresponding to the depth of cut and lead angle. The adjusted cutting speed can be calculated from $V = V_{opt} \times F_f \times F_d$, where V_{opt} is the smaller (*optimum*) of the two speeds from the speed table (from the left side of the column containing the two feed/speed pairs). See the text for examples.

Third, find the feed factor where the step 1 row and step 2 column cross.

Sixth, calculate the final cutting speed.

Calculating Turning RPM (continued)

Step 4:

For this example following the steps in 5a:

Calculate the following ratios: $$\frac{F}{F_{opt}} = \frac{0.012}{0.017} = 0.7$$

And $$\frac{V_{avg}}{V_{opt}} = \frac{790}{585} = 1.35$$

From Table 5a, page 1035, find $F_f = 1.22$ and $F_d = 0.87$

Calculating Turning RPM (continued)

Step 5:

As shown at the bottom of Table 5a,

$$V = V_{opt}\ F_f\ F_d$$

Where V = cutting speed to be used (fpm)

V_{opt} = optimum cutting speed from the table based on material hardness and type of cutter

F_f = feed factor from Table 5a

F_d = depth of cut factor from Table 5a

For this example, V = (585)(1.22)(0.87) = 621 fpm

Calculating Turning RPM (continued)

Step 6:

Finally, calculate the RPM with

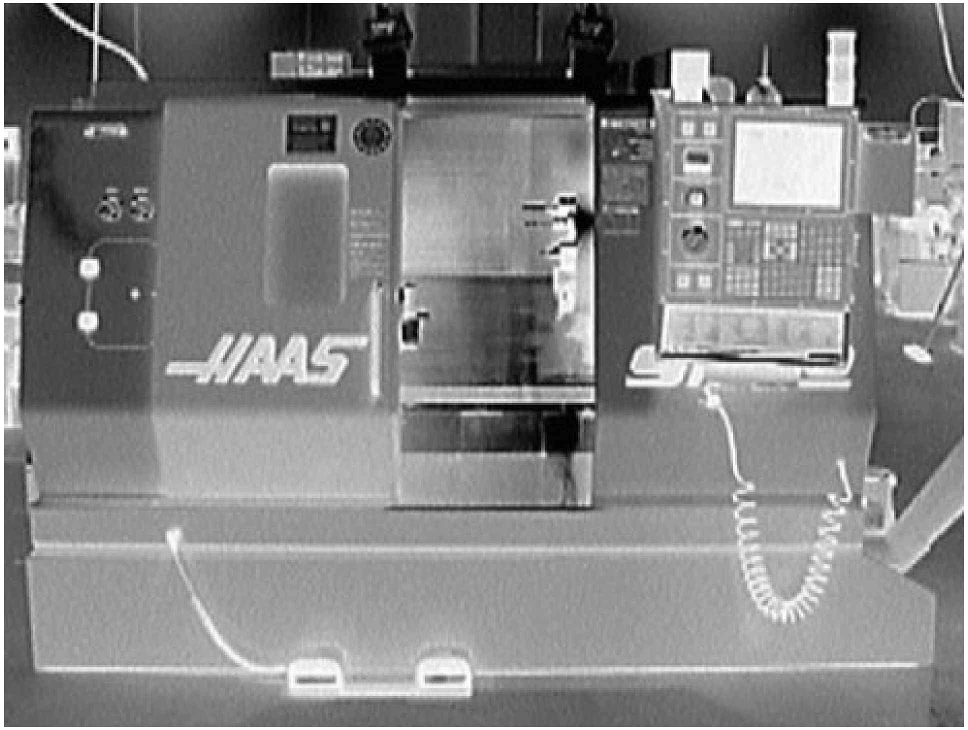
$$N = RPM = \frac{12 \times V}{\pi \times D}$$

For this example:

RPM = 12 x 621 ÷ π ÷ 2.5 = 949 RPM

CNC Applications

Programming Turning Centers



Planning and Programming

To program a CNC machine tool to create a part, you must follow a series of steps to be successful:

1. Examine the part drawing thoroughly and get a rough idea of how you want to proceed.
2. Figure out how to hold the raw material so you can perform as much machining as possible in one setup.
3. Decide what cutters are necessary to perform the various operations.
4. Write down the exact sequence of operations necessary to machine the part, one cutter at a time.
5. Convert your sequence of operations into a program and simulate the program if possible.

What is a block?

- The machine reads the program one line at a time.
- Each line is called a block.
- Blocks do not extend past one line.
- The order of information on a block does not matter:

G0 X3.0 Z1.75 is the same as **Z1.75 G0 X3.0**

- However, most programmers use the following order:

N G X Y Z I J K U V W A B C P Q R F S T M H

- We’ll go over the meaning of each letter as the course progresses.

G and M Codes

The machine operation is divided into two basic types:

- G codes also called preparatory codes
  - tell the machine what type of movement or function should be performed. For example, rapid moves, linear feed moves, arc feed moves, thread cutting, etc.
- M codes also called miscellaneous functions
  - turn the spindle on and off, coolant on and off, etc.

Common Codes

Preparatory G Code	Action	Miscellaneous M Function	Action
G0	Linear rapid traverse positioning move	M3	Spindle forward
G1	Linear feed move	M4	Spindle reverse
G2	CW arc	M5	Spindle off
G3	CCW arc	M8/M9	Coolant on/off
G28	Go home	M30	End of program

Other Codes

Preparatory G Code	Action	Other Functions	Action
G20	Inches	O	Program number
G40	Cancel nose radius compensation	X, Z	Absolute position
G99	IPR feed mode	U, W	Incremental position
G54	First fixture offset	T	Tool number
		S	Spindle Speed

Modal

- Most codes are modal – they stay in effect until something changes them.
- We only program what changes, nothing extra. For example:

Preferred	Works, but poor style
G0 Z3.0 X2.0 G1 Z1.5 F0.012 X2.5	G0 Z3.0 G0 X2.0 Z3.0 G1 X2.0 Z1.5 F0.012 G1 X2.5 Z1.5 F0.012
Easy to read and change!	Difficult to follow, and changes require considerable effort.

Notes on Turning Center G & M Codes

- Most machines only allow one M code per block.
- The capital “Oh” for the program number is the only “Oh” in the program. All others are zeros (0). Be sure you do not mistype.
- The tool code (T) is four digits – the first two for the tool number, the second two for the offset number. They are usually the same.
- All alpha characters must be in uppercase.

Notes on Number Formats

All numbers except zero require a decimal point, otherwise the machine defaults to its resolution. For example:

X3.0	works fine
X3.	works fine
X3	the machines interprets as X0.0003
Z0	works fine
Z0.	works fine

Special Notes for Sending a Turning Center Home

- The G28 code is used to send the machine home.
- G28 requires a move through an intermediate point.
- We generally position the tool clear of the part before sending it home, so the intermediate point is not used.
- To give it a point, we incrementally program a 0 movement like this:
  - G28 U0 which means go home in X incrementally through a point 0 distance from the current location
  - G28 W0 means the same for the Z direction

Program Functions
Fall into just Four (4) Categories

1. Program Start
2. Tool Change
3. Program End
4. Machining Functions

   The first 3 are generally the same for all programs for a given machine.

   Note that they will be different for different machines. **You must know your machine by reading the machine manual!**

Program Functions for the Haas SL-20 Turning Center

- The CNC language is not 100% standard across all machine and control manufacturers.
- Haas machines use fairly generic programming that is similar to most Fanuc compatible machines. Fanuc is probably the most common machine controller.
- Again, **you must know your machine by reading the machine manual!**

Haas SL-20 Program Start

Program	Explanation
%	Starting character for file transfer
O999	Program number set to 999, note the capital “Oh”
G20 G40 G99	Initial conditions
G28 U0	Go home in the X direction
G28 W0	Go home in the Z direction
T0202	Load tool 2 with offset 2
G54	Load the first fixture offset
S4000 M3	Set the spindle to 4000 RPM in the forward direction
G0 Zzzz	Rapid to the first Z location, zzz is the numerical value
Xxxx M8	Rapid to the first X location (G0 is modal), turn the coolant on, xxx is the numerical value
.	
.	
.	Machining moves follow

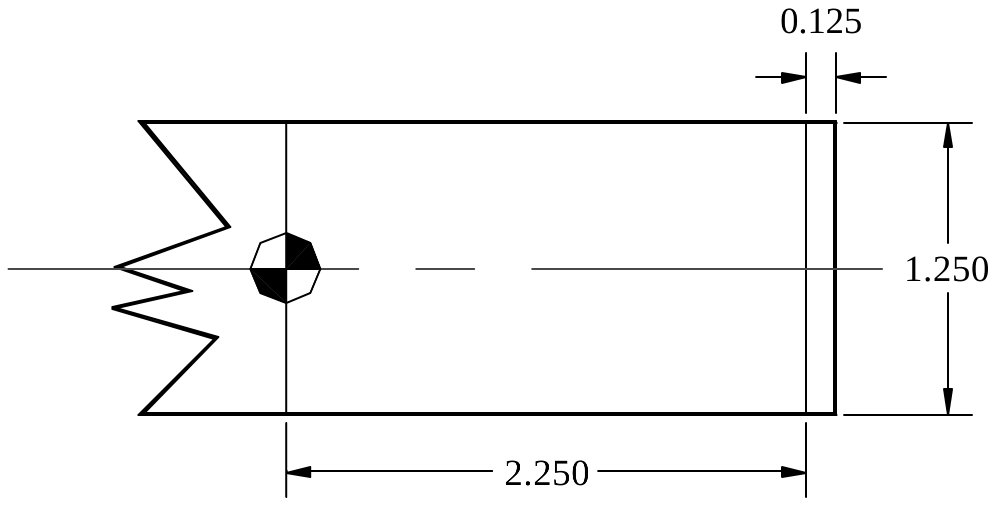
Haas SL-20 Tool Change

Program	Explanation
M9	Turn the coolant off
G28 U0	Go home in the X direction
G28 W0	Go home in the Z direction
T0303	Load the next tool and offset
G54	Load the fixture offset
S3500 M3	Set the spindle speed and direction
G0 Zzzz	Rapid to the first Z location, zzz is the numerical value
Xxxx M8	Rapid to the first X location (G0 is modal), turn the coolant on, xxx is the numerical value
.	
.	
.	Machining moves follow

Haas SL-20 Program End

Program	Explanation
M9	Turn the coolant off
M5	Turn the spindle off
G28 U0	Go home in the X direction
G28 W0	Go home in the Z direction
M30	End of program M code
%	End of file character for file transfer

A Simple Turning Center Program



Material: 1117 CD Steel 175HB

Problem Statement:

Face up to 1/8” (0.125”) off the end of a 1.250” diameter 1117 CD steel bar that is 175 Hb using a hard, coated carbide C shaped insert at 0.004 ipr feed.

Follow Planning and Programming Steps (1-3)

1. Examine drawing
2. How will we hold the raw material – in a collet.
3. Decide what cutters to use – given hard, coated carbide C shaped insert.

Since $f < f_{avg}$, use V_{avg}=1410 fpm giving

$$N = RPM = \frac{12 \times V}{p \times D} = \frac{12 \times 1410}{p \times 1.25} = 4480$$

Since the machine only goes 4500RPM, we'll use 4000RPM to stay a little under the maximum.

Follow Planning and Programming Steps (4)

4. Write down the exact sequence of operations:
    A. Rapid position the cutter in Z
    B. Rapid position the cutter 0.1” away from the part in X
    C. Face to X0. at F0.004
    D. Move away 0.050” in Z
    E. Rapid position 0.1” away from the part in X
    F. Program end.

Follow Planning and Programming Steps (5)

5. Convert the sequence of operations to a program:

Program Start

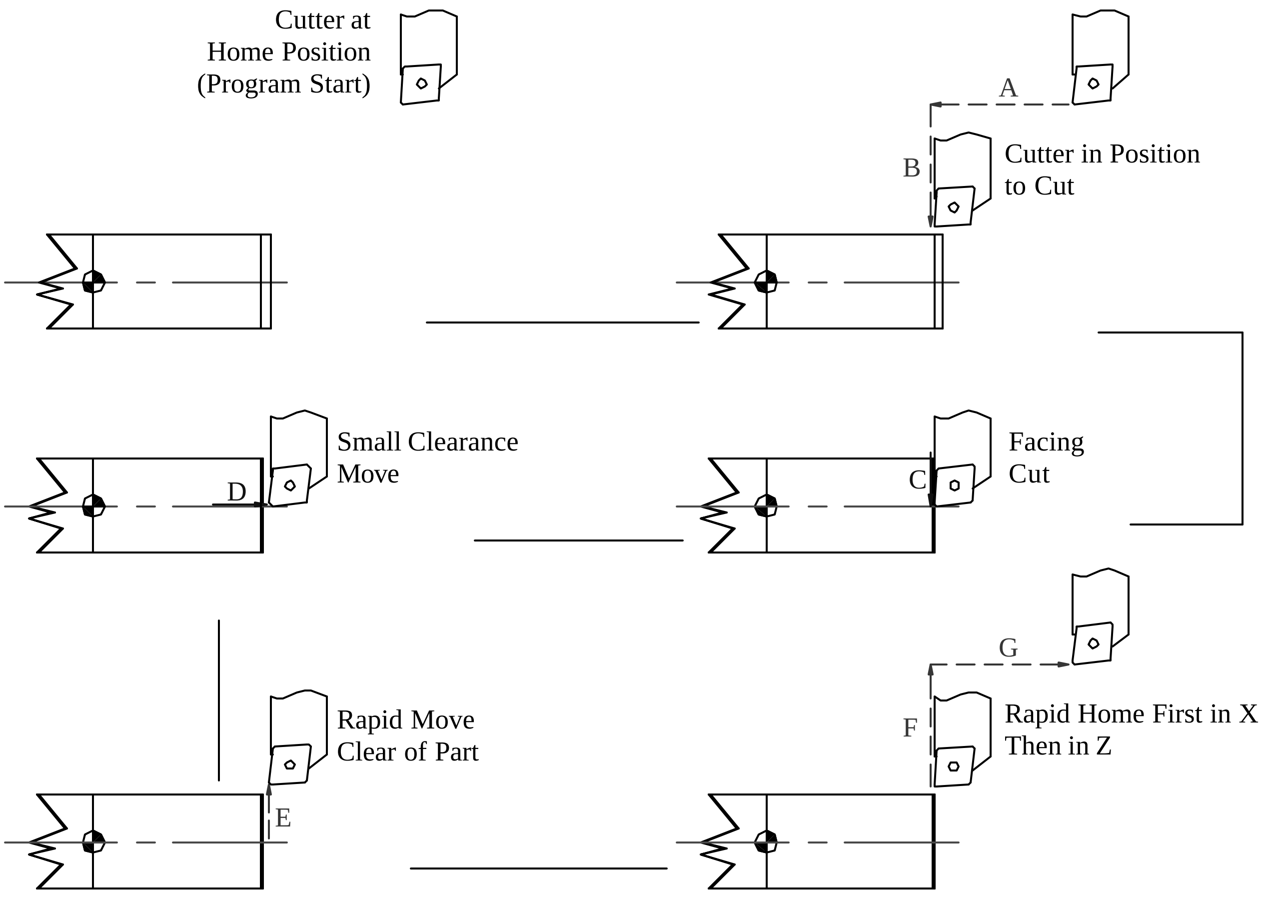
Facing

Program End

The Actual Program

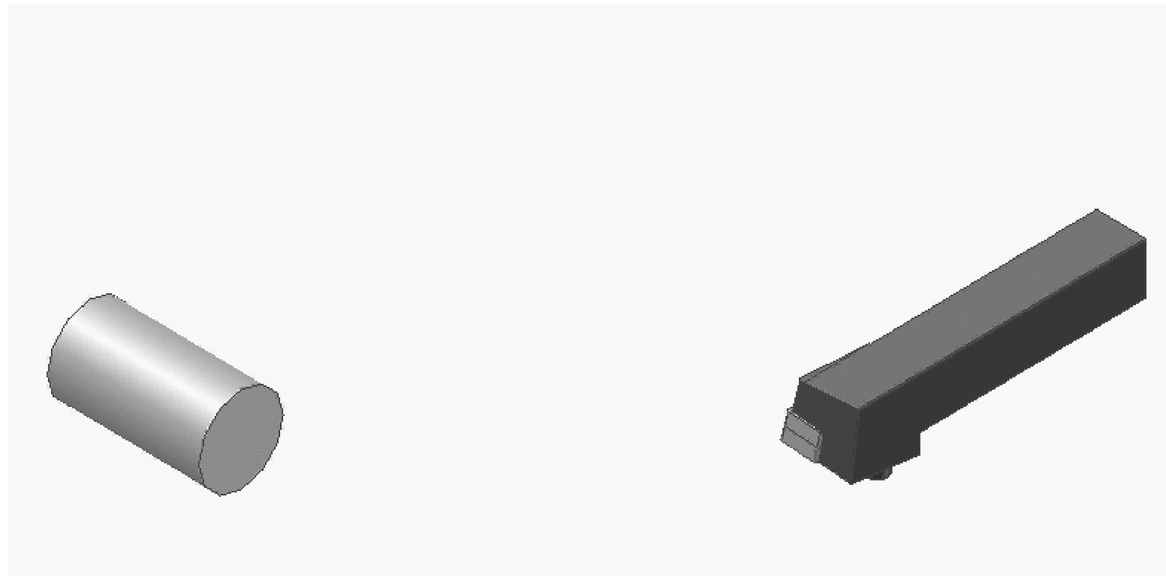
Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0202	
G54	
S4000 M3	
G0 Z2.25	A. Rapid to position in Z
X1.350 M8	B. Rapid 0.1 away from part (0.2 on diameter)
G1 X0 F.006	C. Feed to X is 0 (center of the part)
Z2.3	D. Feed 0.050" away from the face
G0 X1.350	E. Rapid back to 0.1" away from part in X
M9	
M5	
G28 U0	F. Rapid home in X
G28 W0	G. Rapid home in Z
M30	
%	Program End

What the Machine Does



An Animation of the Machine's Movement

Select this link to start the animation.



Notes on the animation:

1. The isometric view orientation is the same as the earlier views of the complete machine with the tailstock to the right.
2. The animation shows the motion of the cutter, but it does not differentiate between rapid and feed moves.
3. The cutter is upside-down because the spindle is going forward (CCW is this view) and the cutter is on the back side of the part.
4. Sorry, I can't make the part rotate or chips fly!

Running a
Program for the First Time

1. Install all cutters in the proper holders.
2. Install the fixture or chuck on the machine and establish the WCS.
3. Set the cutter offsets.
4. Simulate the program on the machine.
5. Slow rapid traverse down as low as possible.
6. Initiate the single step cycle with your hand on the E-stop button at all times.
7. Carefully watch the operations, press the feed hold button to take notes for any corrections.
8. Install a part and go to step 5.

CNC Applications

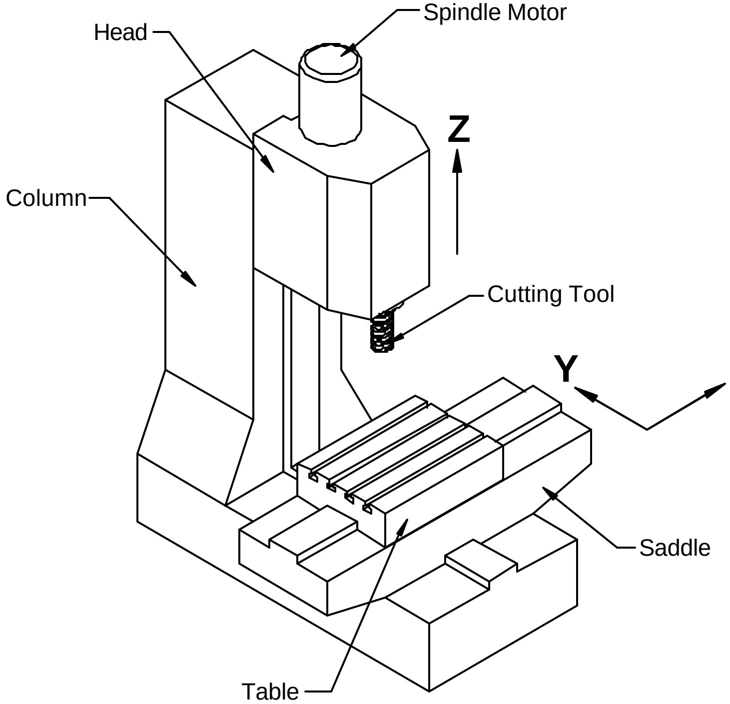
Introduction to Machining Centers

Machining Centers

A machining center is simply a CNC milling machine with an automatic tool changer and an enclosure.

There are a number of different types of machining centers differentiated by the number of programmable axes.

Three Axis Machining Center



With three axes, we can machine one surface of a cube with the end of the cutter and four additional surfaces with the side of the cutter.

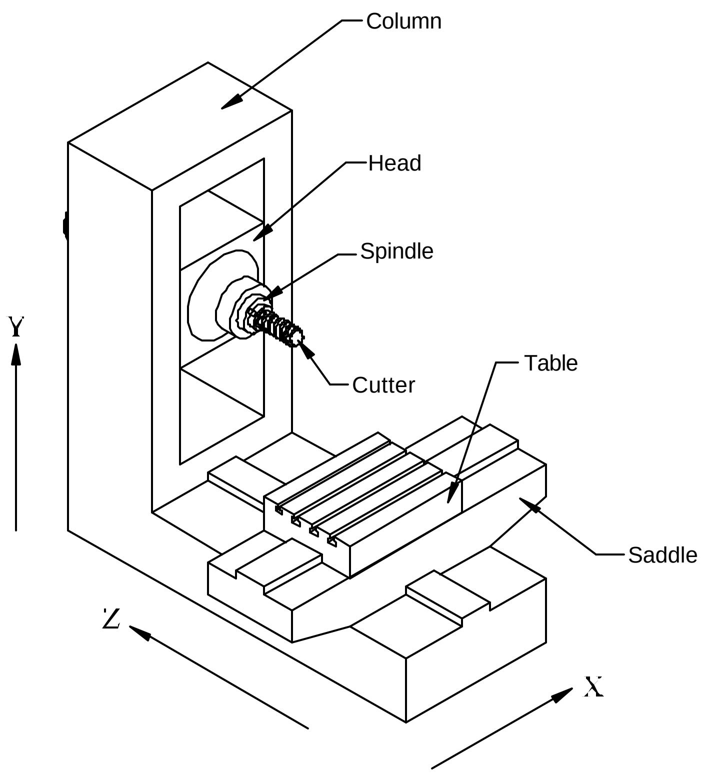
A three axis machining center has programmable X and Y axes in the plane of the table and a Z axis in the spindle's direction.

This is the most basic type of machining center, and they start at about $30,000.

Typically, three axis machines are in the vertical configuration shown here.

Note: tool changer, control, and enclosure not shown.

Horizontal Three Axis Machine



The illustration depicts a three axis horizontal machining center.

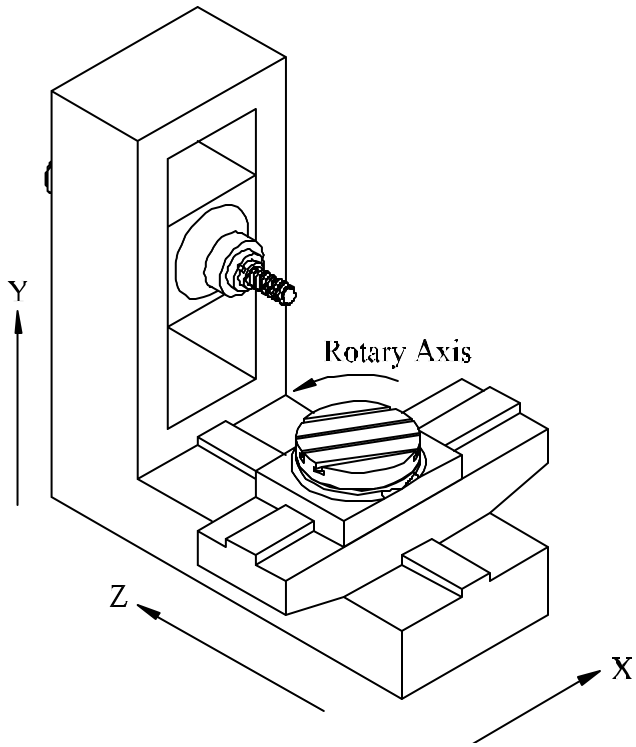
Note the different orientation of the X, Y, and Z axes.

This type of machine starts at about $90,000.

Again, the tool changer, control, and enclosure are not shown.

We can still only machine one surface of a cube with the end of the cutter and four additional surfaces with the side of the cutter.

Four Axis Machining Center



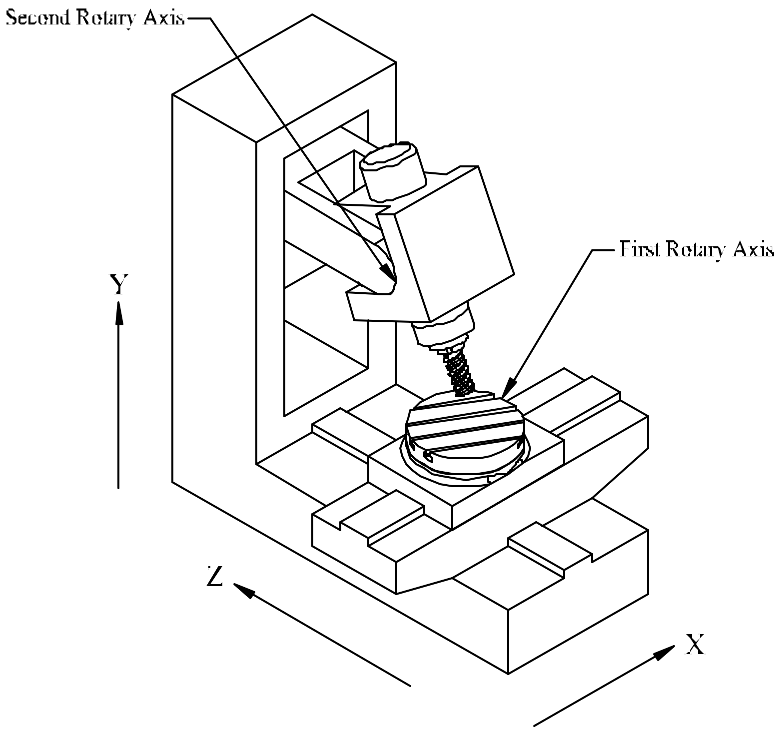
We can machine four surfaces of a cube with the end of the cutter and two additional surfaces with the side of the cutter.

Four axis machining centers are generally horizontal, and the table rotates to create the forth axis. True four axis machines start around $100,000

We can also do this by adding a CNC controlled rotary table to a three axis vertical machine, and this is commonly done for small parts.

Frequently, a fixture called a tombstone (see sketch later) is mounted on the table and many small parts machined at once on a large machine.

Five Axis Machining Center



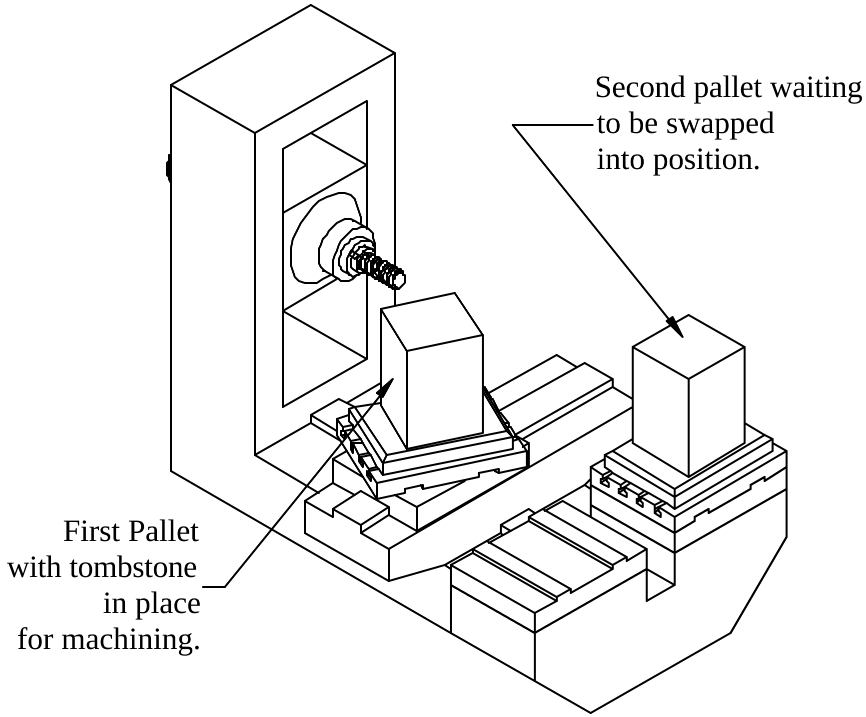
Be prepared to spend about $250,000 for a true five axis machine. Notice that it is very similar to the four axis machine except the spindle rotates from horizontal to vertical.

These machines are used to machine complex parts and molds in the aerospace and automotive industries.

We can machine five sides of a cube with the end of the cutter and six sides with the side of the cutter.

Besides complex geometry, we can often machine a part in one setup on a five axis machine that would require two or more setups in a simpler machine. This results in a more accurate part.

Machining Center with Pallet Changer

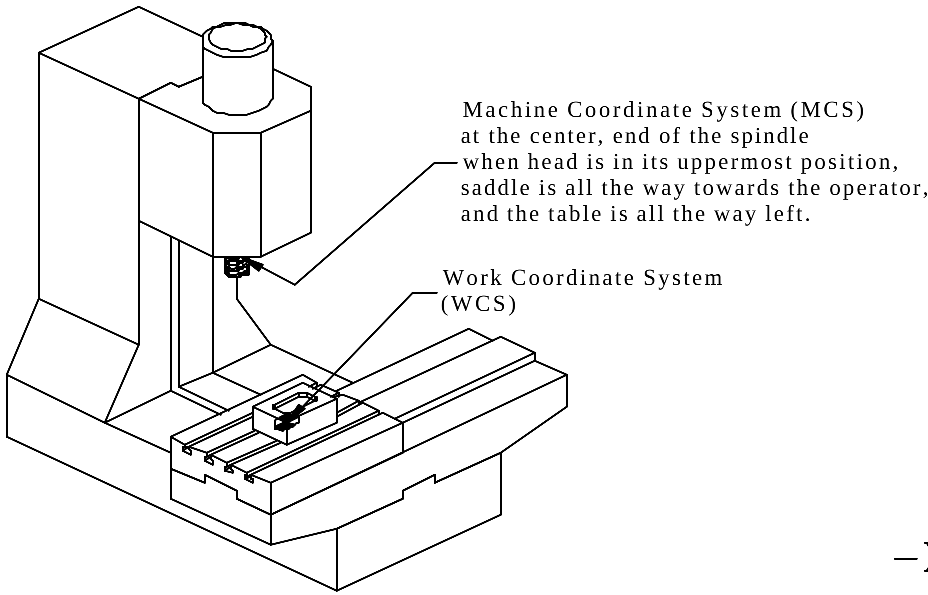


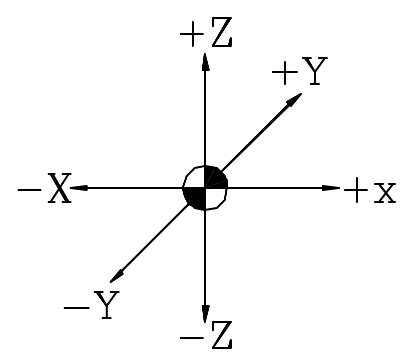
Most machining centers can be fitted with a pallet changer to increase productivity.

On a plain machine, it sits idle while the operator removes the completed parts and loads the fixture with new ones.

With a pallet changer, the operator unloads and reloads one pallet while the machine works on the other. This way, the machine continuously cuts parts.

Machining Center Coordinate System

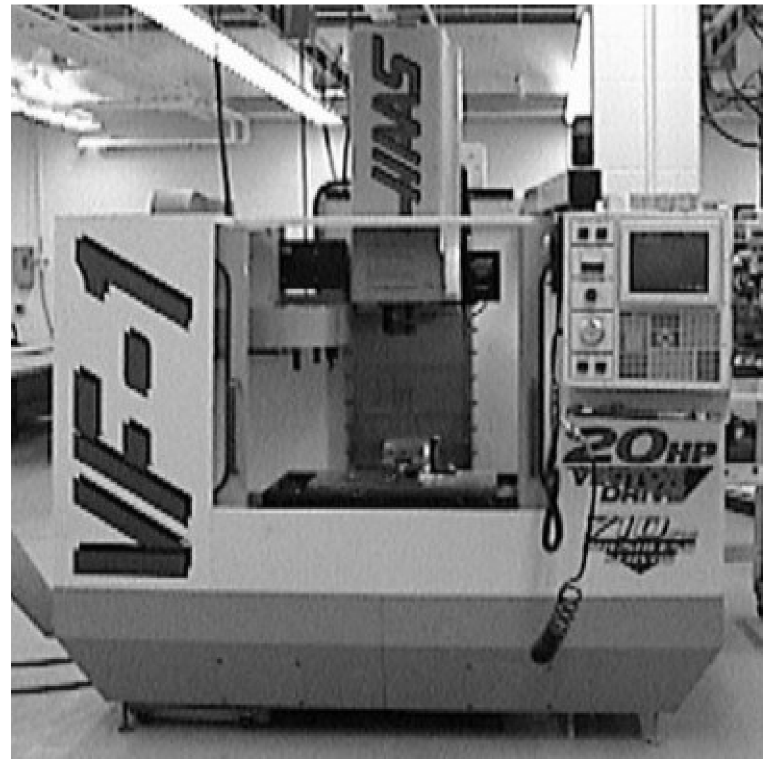




Machining Center
Coordinate System

The Haas VF-1

We will be using a Haas VF-1 three axis machining center with the following specifications: 20hp, 7500 RPM, 710ipm rapid, 300ipm feed, 20 CAT40 tools, 20”x16”x20” travel, 7100lbs!



Milling Machine Coordinate System Concerns

- Regardless of the machine design, **you always program as if you are moving the tool.**
- On most machining centers, the head moves the cutter up and down, so a move in the +Z direction moves the cutter and head up.
- However, most machines move the table in the XY directions. So, a +X move actually moves the table to the left, but the cutter moves in the +X direction relative to the part. Don’t be concerned with this as it is an operation issue, not a programming issue. Remember, **always program as if you are moving the tool.**

Milling Machine Coordinate System Concerns (continued)

- Read the Turning Centers Introduction if you have not already done so.
- As with the lathe, a fixture offset is entered into the machine controller which includes the distance the machine moves from the MCS 0,0,0 to the WCS 0,0,0 position on the part.
- As with the lathe, the programmer picks the WCS on the part. This is more complicated because of the extra Y axis.
- The length of each tool is also entered into the machine control, so the machine compensates for the WCS and the length of the cutter.

Separating the Programmer from the Machine

- As with the lathe, the programmer chooses the WCS on the part, and then programs the cutter movement from there.
- The programmer calls the correct fixture offset and tool length offset numbers in the program (we’ll cover these codes later).
- So, once again, the programmer does not have to be too concerned with any machine specific measurements.

Incremental and Absolute Programming on Machining Centers

Just as with turning centers, machining centers can be programmed with absolute or incremental coordinates, but machining centers use a different format:

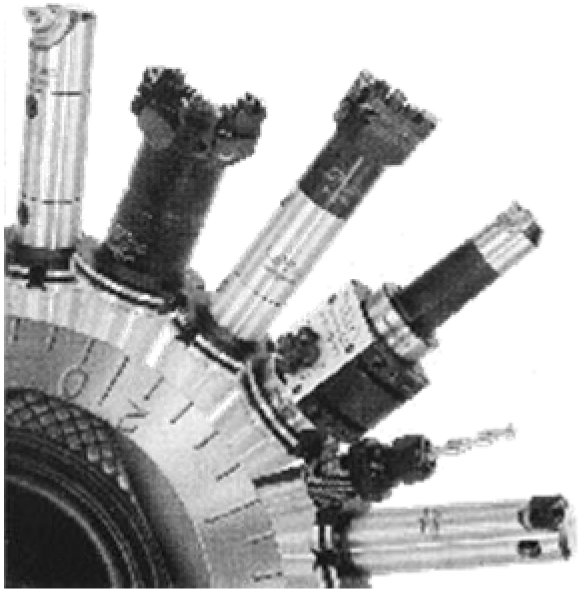
Turning Centers		Machining Centers	
Absolute 0,0 start	Incremental 0,0 start	Absolute 0,0,0 start	Incremental 0,0,0 start
G0 X3. Z2. X3.5	G0 U3. W2. U0.5	G90 G0 X3. Y2.0 X3.5	G91 G0 X3. Y2. X0.5
Notice the use of X,Z for absolute and U,W for incremental.		Notice that G90 signifies absolute coordinates and G91 signifies incremental, and both use X,Y,Z.	

Incremental vs. Absolute Programming

- As with turning centers, most machining center programming will be done in absolute mode.
- Editing for program changes is much easier in absolute mode, and absolute programs are much easier to follow.
- Certain repetitive operations such as drilling multiple holes lend themselves to incremental programming, and we will cover this later in the course.

CNC Applications

Tooling for Machining Centers



Cutting Tools

ENDMILLS

Most machining centers use some form of HSS or carbide insert endmill as the basic cutting tool.

Insert endmills cut many times faster than HSS, but the HSS endmills leave a better finish when side cutting.

Photo courtesy ISCAR.

Cutting Tools (continued)

Facemills flatten large surfaces quickly and with an excellent finish. Notice the engine block being finished in one pass with a large cutter.

Photo courtesy ISCAR.

Cutting Tools (continued)

Ball endmills (both HSS and insert) are used for a variety of profiling operations such as the mold shown in the picture.

Slitting and side cutters are used when deep, narrow slots must be cut.

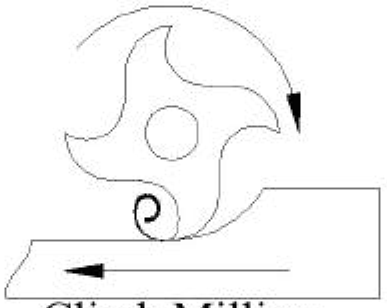
Photos courtesy ISCAR.

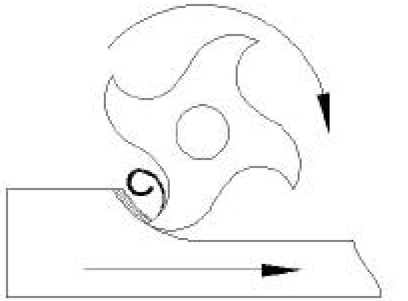
Milling Feed Direction

Remember, all CNC machines are equipped with ball screws to minimize slop when changing feed directions. The other advantage to ball screws is they allow climb milling instead of conventional milling as done on most manual machines.

Climb milling has many advantages including better surface finish, longer tool life, and the cutter deflects away from the work rather than into it.

Always climb mill on a CNC machining center!

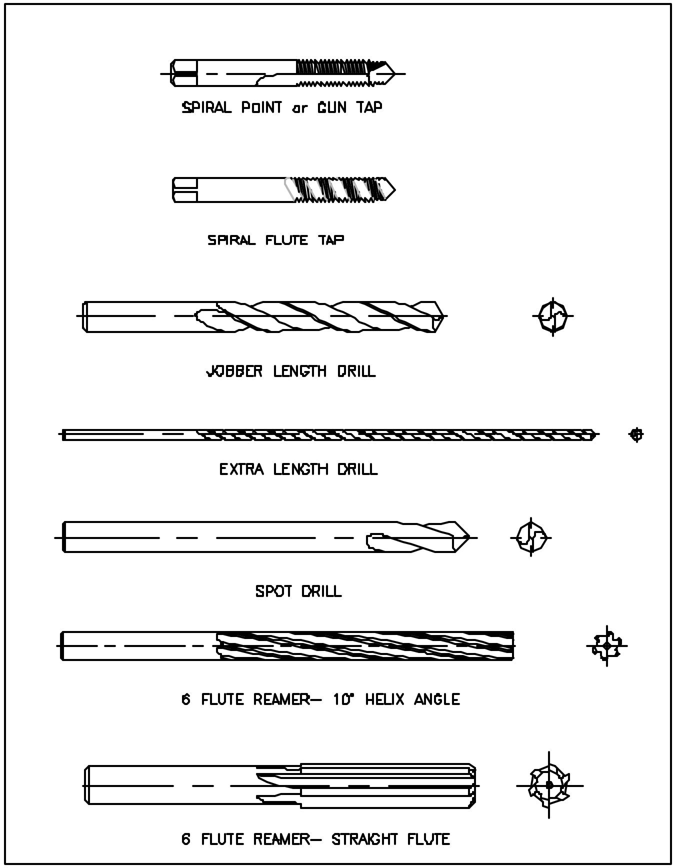

Climb Milling


Conventional Milling

Drills, Taps, and Reamers

Common HSS tools such as drills, taps, and reamers are commonly used on CNC machining centers. Note that a spot drill is used instead of a centerdrill. Also, spiral point or gun taps are used for through holes and spiral flute for blind holes. Rarely are hand taps used on a machining center.

Drawings courtesy Precision Twist Drill.



Tool Holders

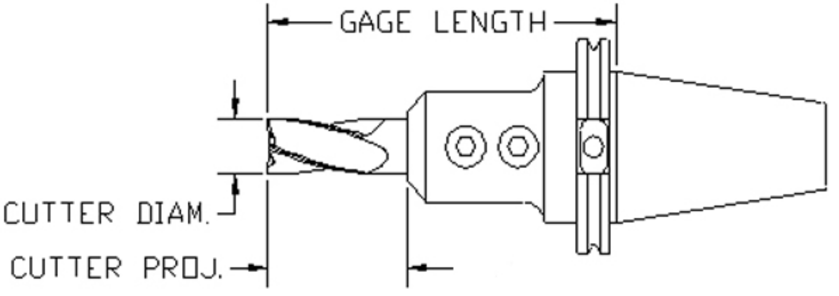
All cutting tools must be held in a holder that fits in the spindle. These include end mill holders (shown), collet holders, face mill adapters, etc.

Most machines in the USA use a CAT taper which is a modified NST 30, 40, or 50 taper that uses a pull stud and a groove in the flange. The machine pulls on the pull stud to hold the holder in the spindle, and the groove in the flange gives the automatic tool changer something to hold onto.

HSK tool holders were designed a number of years ago as an improvement to CAT tapers, but they are gaining acceptance slowly.

Photo courtesy Fitz-Rite

The gage length shown in the drawing is entered in the machine control as the tool length. The machine then compensates for the length.



Fixtures

Fixtures include anything that holds the work on the machining center table.

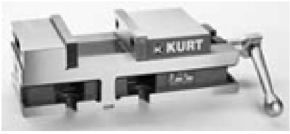
Holding parts on a machining center tends to be much more difficult than on a turning center.

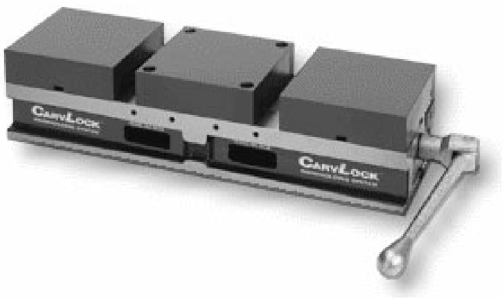
The simplest fixture is just a vise as shown in the top photo.

The next photo shows a double vise with machinable jaws to hold odd shaped pieces.

The tombstone shown in the lower photo has a double vise on each face for use on a four or five axis machining center.

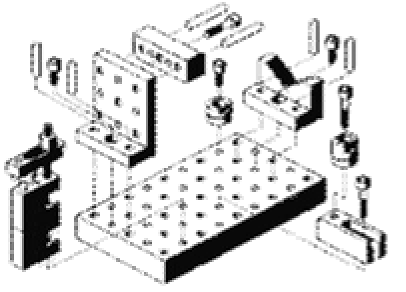
Photos courtesy Kurt Manufacturing.





More Fixtures

When parts can’t be held in a vise, a custom fixture must be used. Fixtures for high production parts are often custom designed and manufactured at great expense.

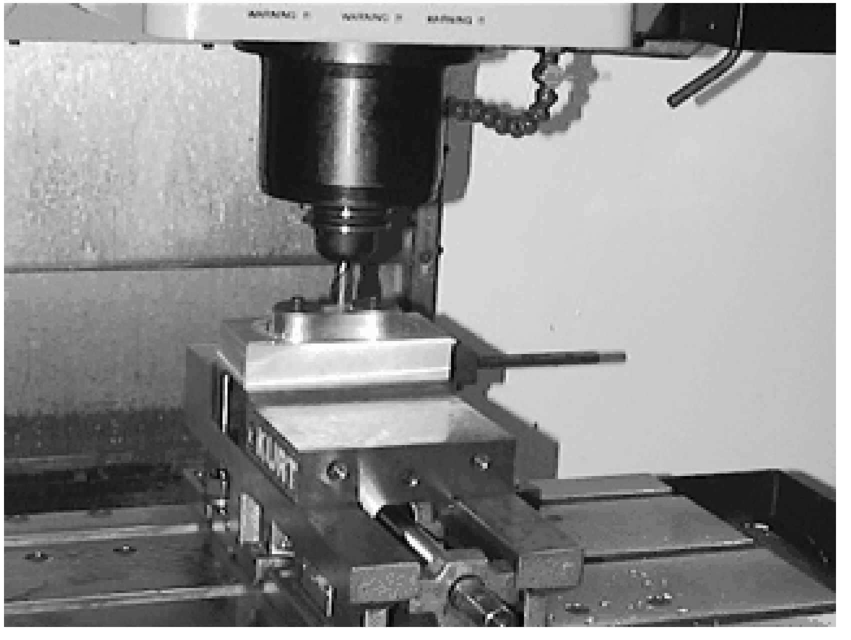


For small runs of odd-shaped parts, many manufacturers have turned to modular fixturing. As shown in the upper drawing, modular fixturing consists of many precision ground pieces that fit together to hold all sorts of parts as shown in the lower photo.

Drawing and photo courtesy Bluco.

CNC Applications

Speed and Feed Calculations for Milling



Calculating RPM for Milling Operations with HSS Cutters

We use the same basic formula as for turning, except D is now the cutter diameter:

$$N = RPM = \frac{12 \times V}{p \times D}$$

Tables 10 through 16 list milling data. Table 6 must be used for copper alloys.

Example Milling RPM Calculation

- Mill 4140 steel with a Brinell hardness of 200 with a ½" HSS endmill.
- From Table 11, page 1047, we find V = 75 fpm, so:

$$RPM = 12 \times 75 \div p \div 0.5 = 573$$

Feed Rates for Milling

- Feed for milling cutters is usually tabulated as inches per tooth (ipt), but feed rates on milling machines are programmed in inches per minute (ipm). The equation on page 1041 in the *Handbook* is given as

  $f_m = f_t \, n_t \, N$
- Where
  - f_m is the feed rate in ipm we want to set the mill at.
  - f_t is the feed rate in inches per tooth, ipt.
  - n_t is the number of teeth on the cutter we are going to use.
  - N is the RPM we already calculated.
- See Table 15a, pages 1054-1055, for values of f_t with HSS cutters.

Example Milling Feed Rate Calculation

Mill 4140 steel with a Brinell hardness of 200 with a ½” 4 flute HSS endmill and a ¼” depth of cut.

We already calculated the RPM at 573,

From Table 15a, $f_t = 0.001$“

$$f_m = 0.001 \times 4 \times 573 = 2.3 \text{ IPM}$$

Feed Rate Concerns

- Feed rates for facemill and slotting cutters vary widely.
- Slow feed rates give a better finish, but sometimes this actually dulls the cutter faster than a more rapid feed rate.
- Data for carbide insert milling cutters should be obtained from the insert manufacturer. Unlike lathe cutters which are fairly standard, milling cutters vary widely between manufacturers, so use your manufacturer’s data.

Calculating RPM for Drilling Operations with HSS Cutters

- The RPM formula for drilling is the same as for turning and milling, except D is now the drill diameter.

- Tables 17-23 lists data for drilling, reaming, and threading.

- Note: deeper holes require slower cutting speeds because the coolant cannot reach the cutting edge effectively.

- Feed rates for drilling are given in a paragraph on page 1060. Note that the values are given in inches/revolution (ipr). If you need ipm, multiply ipr by RPM.

Drilling RPM and Feed Calculation Example

- Drill cold drawn free cutting brass, C36000, with a 1” drill. From Table 23, page 1072, we find V = 175 fpm, so:

$$RPM = 12 \times 175 \div p \div 1 = 668$$

- From page 1060, the feed would be between 0.007 and 0.015 ipr.
- To find ipm, use ipm = Feed x RPM = 0.015 x 668 = 10 ipm

CNC Applications

Programming Machining Centers

Planning and Programming

Just as with the turning center, you must follow a series of steps to create a successful program:

1. Examine the part drawing thoroughly and get a rough idea of how you want to proceed.
2. Figure out how to hold the raw material so you can perform as much machining as possible in one setup.
3. Decide what cutters are necessary to perform the various operations. This is more critical on machining centers because the holder and fixture can interfere with the work.
4. Write down the exact sequence of operations necessary to machine the part, one cutter at a time.
5. Convert your sequence of operations into a program and simulate the program if possible.

G and M Codes

Just as with turning centers, machining centers have two basic types of codes:

- G codes also called preparatory codes
  - tell the machine what type of movement or function should be performed. For example, rapid moves, linear feed moves, arc feed moves, thread cutting, etc.
- M codes also called miscellaneous functions
  - turn the spindle on and off, coolant on and off, etc.

We already noted the G90/G91 for absolute and incremental programming. Another code unique to machining centers is M6 – tool change.

Common Codes

Preparatory G Code	Action	Miscellaneous M Function	Action
G0	Linear rapid traverse positioning move	M3/M4	Spindle forward/Spindle reverse
G1	Linear feed move	M5	Spindle off
G2	CW arc	M0	Program stop
G3	CCW arc	M8/M9	Coolant on/off
G28	Go home	M30	End of program
G90/G91	Abs./Incr. Programming	M6	Tool Change

Other Codes

Preparatory G Code	Action	Other Functions	Action
G20	Inches	O	Program number
G40	Cancel nose radius compensation	X, Y, Z	Absolute position
G99	IPR feed mode	I, J	Arc Vectors
G54	First fixture offset	T, H	Tool Number, Length Offset
G80	Cancel hole cycle	S	Spindle Speed

Modal

- Most codes are still modal – they stay in effect until something changes them.
- We only program what changes, nothing extra. For example:

Preferred	Works, but poor style
G1 Z-.8 F20.0 Y2.4 G0 Z0.1 Y-0.4	G1 Z-.8 F20.0 G1 Y2.4 Z-0.8 G0 Y2.4 Z0.1 G0 Z0.1 Y-0.4
Easy to read and change!	Difficult to follow, and changes require considerable effort.

Notes on Machining Center G & M Codes

- Most machines only allow one M code per block.
- The capital “Oh” for the program number is the only “Oh” in the program. All others are zeros (0). Be sure you do not mistype.
- Unlike the lathe, the tool code (T) is two digits, we’ll cover how to handle the length offset shortly.
- All alpha characters must be in uppercase.
- Don’t forget to put decimal points on all numbers except 0’s! Remember, the machine thinks X3 really is X0.0003.

Special Notes for Sending a Machining Center Home

- Just as with the lathe, the G28 code is used to send the machine home.
- G28 still requires a move through an intermediate point.
- We generally position the tool clear of the part before sending it home, so the intermediate point is not used.
- To give it a point, we incrementally program a 0 movement like this:
  – G91 G28 Z0 which means go home in Z incrementally through a point 0 distance from the current location
  – G91 G28 X0 means the same for the X direction
  – G91 G28 Y0 means the same for the Y direction
- Often, with a machining center, we only send it home in Z or in Z and Y.

Handling Tools on a Machining Center

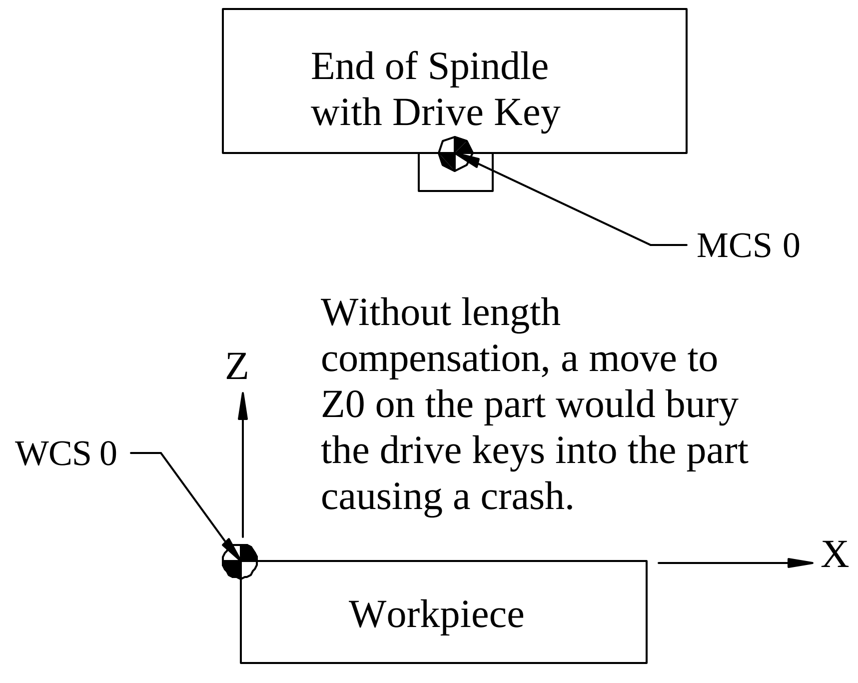
Code	Function
Ttt	Call up tool number tt
M6	Do the tool change
G43	Load the length offset
Htt	Offset number tt
G49	Cancel length offset

Changing tools is very machine specific, so be sure you know your machine!

Generally, the five codes shown in the table load the tool and the length offset as we'll demonstrate on the next few slides.

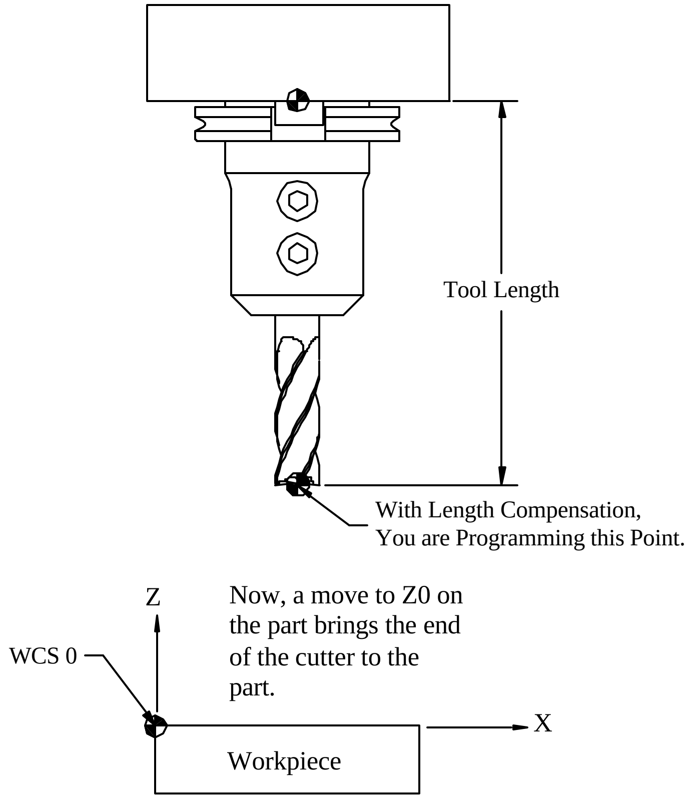
Cutter Length Offset on a Machining Center (1)

Recall that in the Z direction, the MCS is at the end of the spindle, and the fixture offset measures the distance from the MCS to the WCS so the machine can compensate for the location of the part.



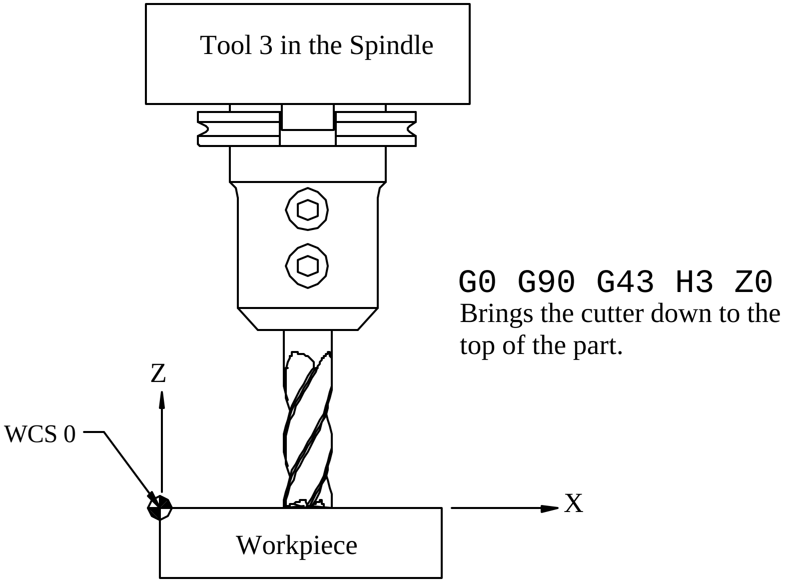
Cutter Length Offset on a Machining Center (2)

Length compensation subtracts the tool length from the distance between the MCS and WCS in the Z direction, so now the programmer is programming the center, end of the cutter.



Cutter Length Offset on a Machining Center (3)

With proper length compensation as shown, the programmer can safely program in the WCS with little regard to the cutter except to insure that the flutes are long enough and the toolholder does not interfere with the workpiece or with the fixture.



Program Functions fall into just four (4) Categories

1. Program Start
2. Toolchange
3. Program End
4. Machining Functions

The first 3 are generally the same for all programs for a given machine.

Note that they will be different for different machines. **You must know your machine by reading the machine manual!**

Program Functions for the Haas VF-1 Machining Center

- Remember, the CNC language is not 100% standard across all machine and control manufacturers.
- Haas machines use fairly generic programming that is similar to most Fanuc compatible machines. Be especially careful of tool changes and sending the machine home!
- Again, **you must know your machine by reading the machine manual!**

Haas VF-1 Program Start

Program	Explanation
%	Starting character for file transfer
O999	Program number set to 999, note the capital “Oh”
G20 G40 G49 G80 G99	Initial conditions
G91 G28 Z0	Incrementally go home in the Z direction
G90	Absolute positioning
T1M6	Call Tool 1 and do the toolchange
S3000 M3	Set the spindle to 3000 RPM, forward direction
G0 G90 G54 X-0.4 Y-0.4	Go to first X,Y position in the WCS
G43 H1 Z0.1 M8	Load the length offset, move to Z0.1, coolant on
. . .	Machining moves follow

Haas VF-1 Program Toolchange

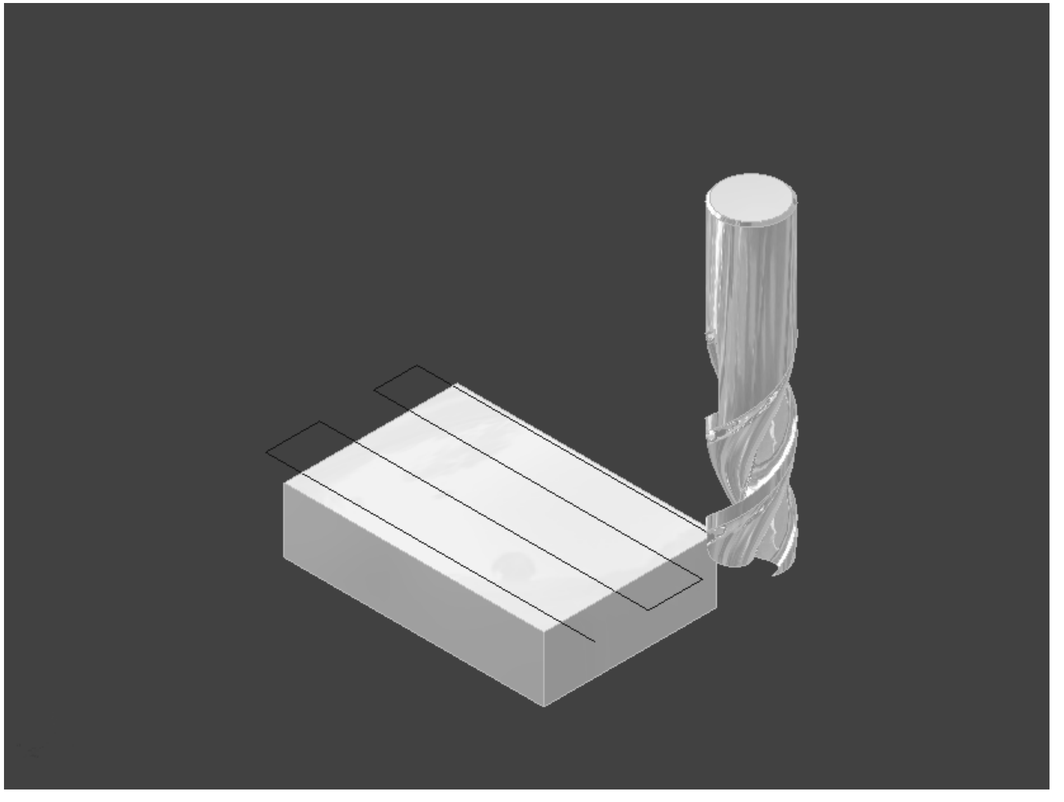
Program	Explanation
M9	Turn the coolant off
M5	Turn the spindle off
G49	Cancel tool length compensation
G91 G28 Z0	Incrementally go home in the Z direction
G90	Absolute positioning
T2M6	Call Tool 2 and do the toolchange
S4500 M3	Set the spindle to 4500 RPM, forward direction
G0 G90 G54 X0.75 Y1.0	Go to first X,Y position in the WCS
G43 H2 Z0.1 M8	Load the length offset, move to Z0.1, coolant on
.	
.	Machining moves follow
.	

Haas VF-1 Program End

Program	Explanation
M9	Turn the coolant off
M5	Turn the spindle off
G49	Cancel tool length compensation
G91 G28 Z0	Incrementally go home in the Z direction
G28 Y0	Home in the Y direction to make unloading the part easier
G90	Absolute positioning
M30	End of program M code
%	End of file character for file transfer

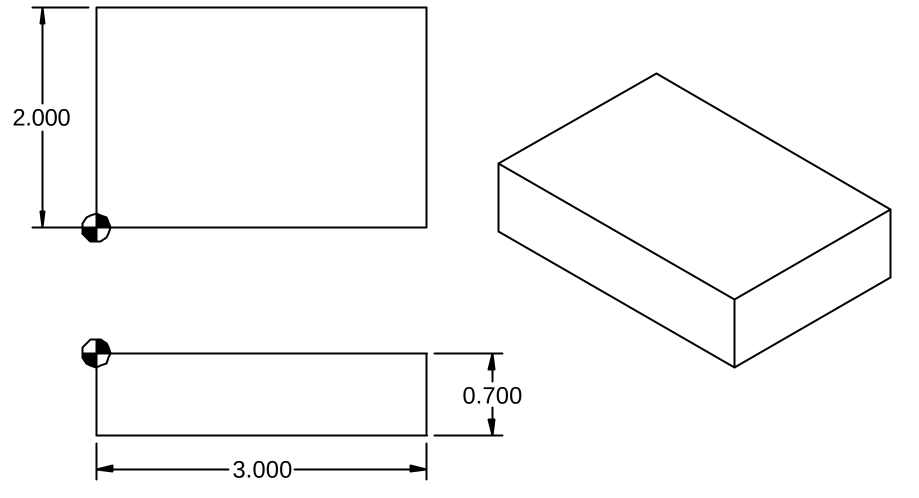
CNC Applications

Machining Center Example #1



Problem Statement

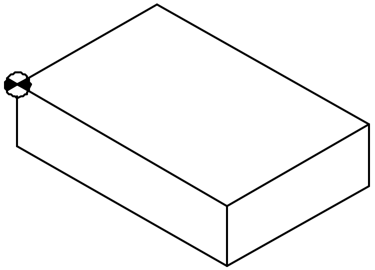
Machine the length and thickness of the part shown below. The part is made from ¾”x2” 6061 CD aluminum which is saw cut to approximately 3 1/8” length. Perform all machining with a 2 flute, ¾” diameter, HSS endmill which is tool 1 on the machine.



Planning and Programming (1)

1. Examine the part drawing thoroughly and get a rough idea of how you want to proceed.

   A. Pick the WCS in the lower left corner of the part on the finished upper surface:
   B. Machine one end with the part against a stop.
   C. Program stop, flip the part, and machine the 3” length.
   D. Machine 0.050” off the top of the part leaving the final 0.700” thickness.



Planning and Programming (2-3)

2. How will we hold the part? In a 6” vise up on 1/8” wide parallels that hold the part only 3/8” into the vise jaws. A stop on the right positions the part.
3. Decide what cutters to use – given a ¾” diameter 2 flute HSS endmill. From the Machinery’s Handbook, we note that this endmill has 1 5/16” of useable flute length.

Planning and Programming (3 cont.)

3. For the endmill, we find from Table 10 that V=600fpm and from Table 15a f_t=0.004ipt:

$$N = RPM = \frac{12 \times V}{p \times D} = \frac{12 \times 600}{p \times 0.75} = 3056$$

$f_m = f_t \, n_t \, N = 0.004 \times 2 \times 3056 = 24$ ipm

Planning and Programming (4)

4. Write down the exact sequence of operations:

   A. Rapid position cutter 1/16” to the left and clear in Y
   B. Feed to depth, face left end of the part, rapid up
   C. Move home in Y,Z and then flip the part
   D. Repeat A and B except for final X position
   E. Face the extra 0.050” off the top of the part
   F. Program end.

Planning and Programming (5)

5. Convert the sequence of operations to a program:

Program Start

Machine Left End

Flip Part

Machine to Length

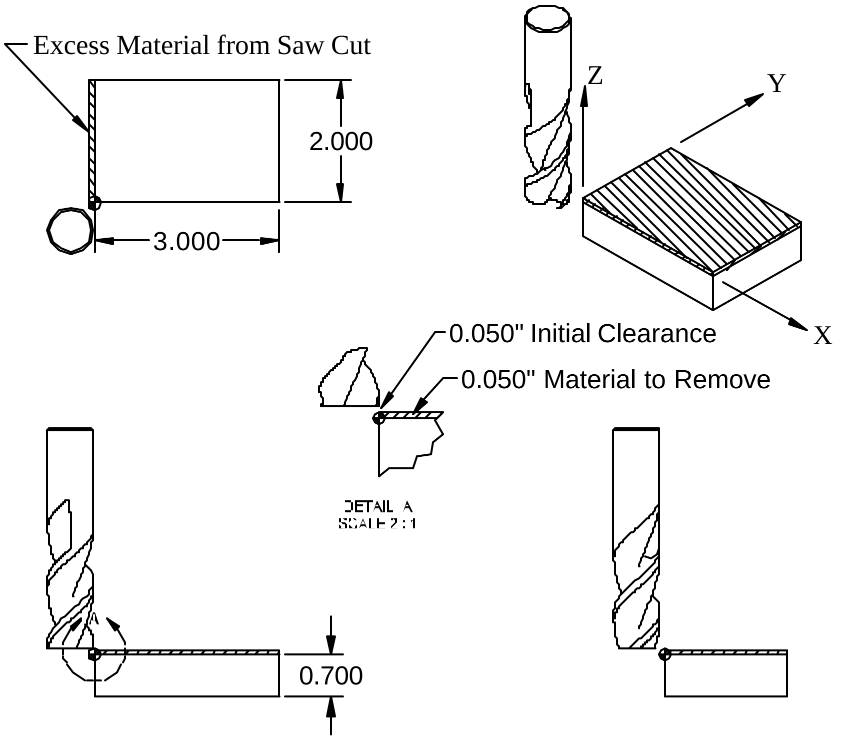
Machine Thickness

Program End

An Overview of the Process

Notes:

1. Both ends of the part are saw cut. We will make one cut leaving 1/16" excess stock and then flip the part and remove the excess length.
2. We will initially position at Z0.1, but realize that we chose the Z=0 plane on the top of the FINISHED part, so we will only have 0.050" clearance as the detail view shows.
3. Remember, program as if the cutter moves in all 3 directions, even though the part moves in the X,Y directions.
4. Remember, we are programming the center of the cutter. All cuts must be offset by the cutter radius,

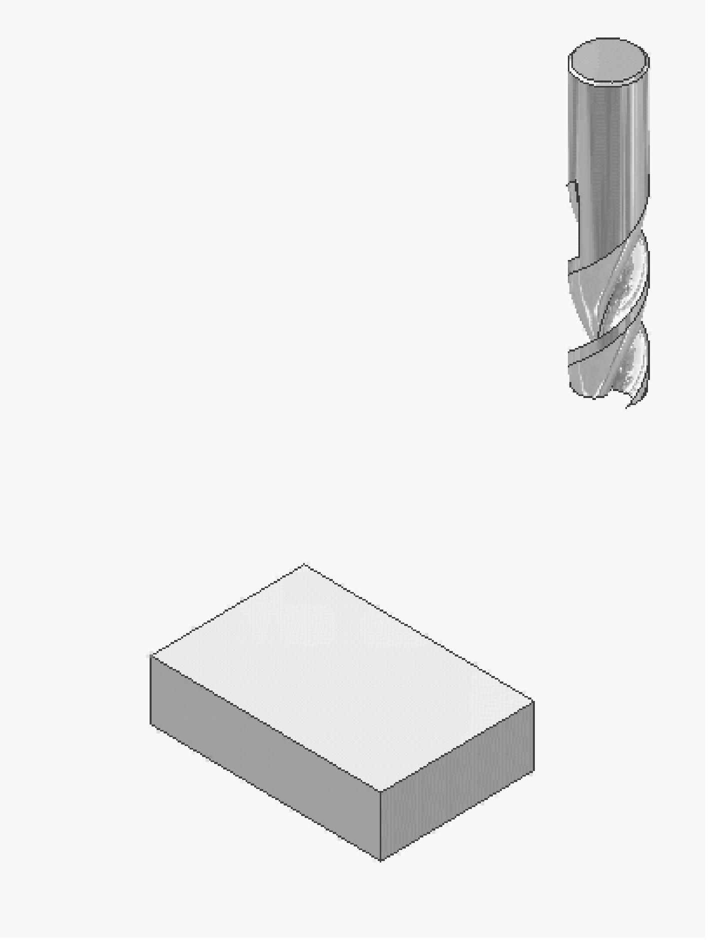


The First Portion of the Program

Program Codes	Action
% O999 G20 G40 G49 G80 G99 G91 G28 Z0 G90	Program Start
T1 M6	Load tool 1, ¾" HSS endmill
S3056 M3	Set the spindle RPM and direction
G0 G90 G54 X-0.437 Y-0.4	Go to initial position in the WCS using fixture offset G54
G43 H1 Z0.1 M8	Rapid to clearance with length compensation, coolant on
	Start of Machining
G1 Z-0.8 F24.	Feed to depth – below part so a large burr is not left
Y2.1	Cut the end of the part until clear in Y
G0 Z0.1	Rapid to clearance plane
M9	Coolant off
M5	Spindle off
G91 G28 Z0	Go home in Z first to avoid hitting anything that sticks up
G28 Y0	Then, go home in Y
G90	Absolute positioning again to cancel the G91
M0	Program stop

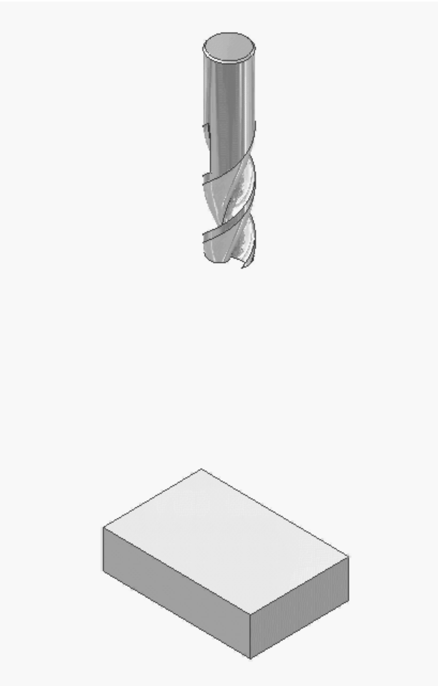
What the Machine Does

Select this link to start the animation.



Continue with Second Length Cut

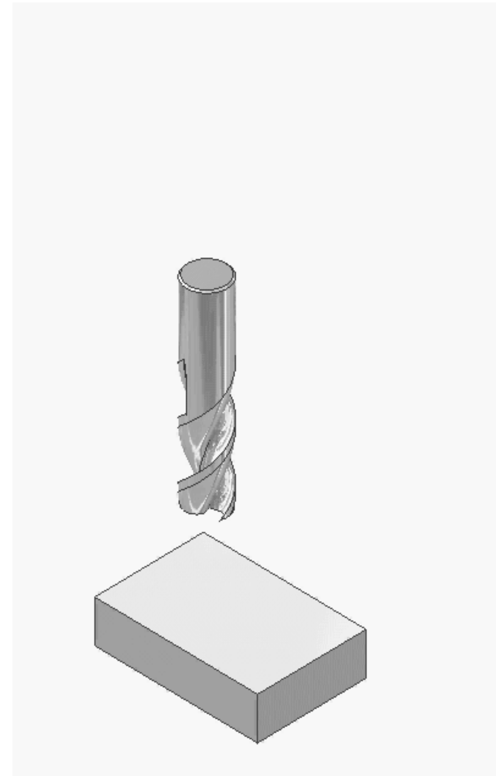
Program Codes	Action
M3 G0 X-0.375 Y-0.4 Z0.1 M8 G1 Z-0.8 Y2.1 G0 Z0	Turn the spindle on, it will use 3056RPM Locate for the second cut to length Rapid down to clearance plane Feed below the part Machine the end, it will use 24IPM Rapid to finished height



Select this link to start the animation.

Face the Top

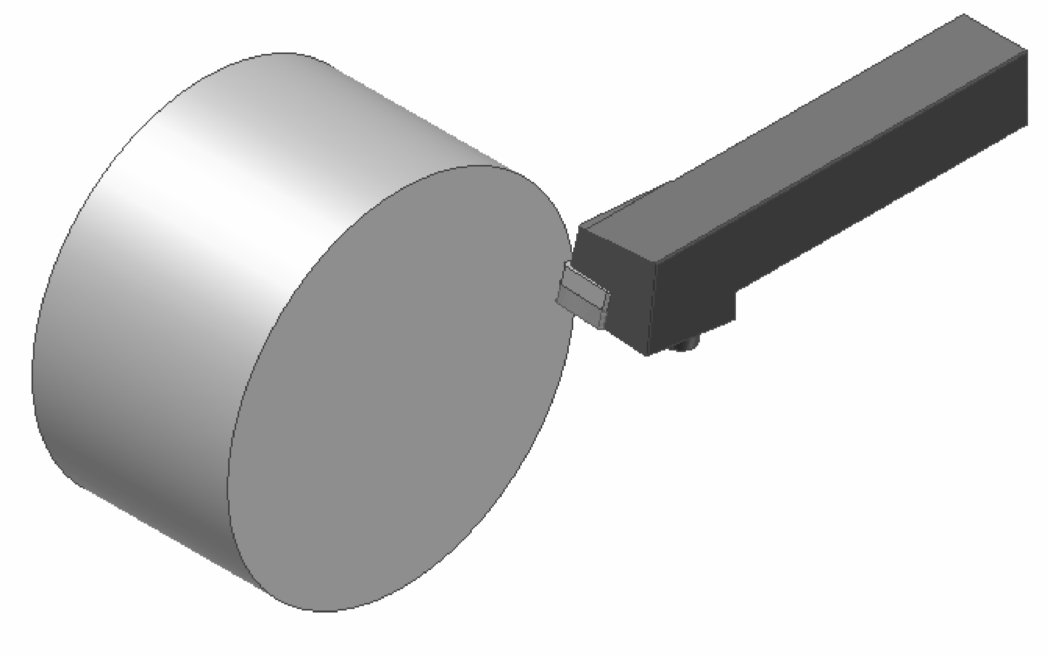
Program Codes	Action
Y1.75	Position for first pass
G1 X3.4	Repeat passes
G0 Y1.125	.
G1 X-0.4	
G0 Y0.5	.
G1 X3.4	
G0 Y0	.
G1 X-0.4	
M9	Coolant off
M5	Spindle off
G49	Cancel length
G91 G28 Z0	compensation
G28 Y0	Go home in Z first
G90	Then go home in Y
M30	Absolute mode
%	End of program
	End of file



Select this link to
Start the animation.

CNC Applications

Constant Cutting Speed (CSS) for Turning Centers



Constant Cutting Speed (CSS)

From our previous lessons, we know to calculate RPM with this formula:

$$N = RPM = \frac{12 \times V}{p \times D}$$

Using some simple algebra, we can rearrange that equation and solve for cutting speed (V):

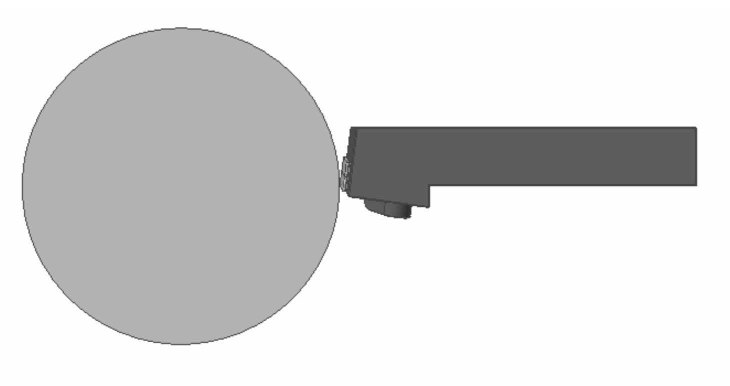
$$V = sfm = \frac{p \times D \times N}{12}$$

Compare
Cutting Speed when Facing

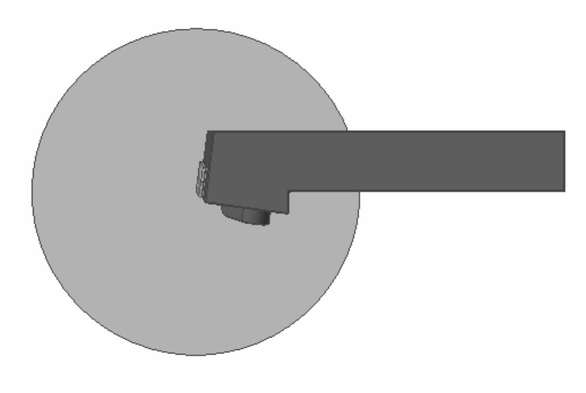
We’ll face a 4” diameter part starting with V=600fpm. We calculate RPM as:

$$N = \frac{12 \times 600}{p \times 4} = 573 RPM$$

Now, we’ll work backwards and calculate the cutting speed (V) at the maximum diameter and towards the center at 0.25” diameter given a constant RPM=573.



$$V = \frac{p \times 4 \times 573}{12} = 600\, fpm$$



$$V = \frac{p \times 0.25 \times 573}{12} = 37.5\, fpm$$

What does this mean?

- Any particular cutting speed is valid at only one RPM and diameter.
- As we decrease diameter at a constant RPM, cutting speed falls until it is zero when the cutter is at the part centerline.
- This decrease in cutting speed results in poor surface finish and shorter cutter life since hard cutters generally perform better at higher cutting speeds.

How do we fix the problem?

- Use a function called Constant Cutting Speed (CSS).
- CSS causes the machine to adjust RPM based on the cutter's X diameter to hold a particular cutting speed.
- Only uses three new G codes:

Code	Function
G96	Sets the cutting speed to the value specified by S.
G97	Sets the RPM to the value specified by S.
G50	Limits the RPM to the value specified by S. Used with G96.

A Sample CSS Program

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0202	
G54	
G50 S4000	Limit the RPM to 4000 (slightly less than the machine max).
G96 S600 M3	Change to CSS, 600 fpm, machine sets RPM based on cutter position.
.	Machining
.	
.	
G97 S1800	Set the RPM to 1800 constant (won't change with cutter position).
.	More machining
.	
.	
G96	Change to CSS, 600 fpm (modal), machine sets RPM.

CSS Comments

- Most machines default to G97, RPM mode, but both G96 and G97 are modal. However, it is a good idea to put the correct one in your program at each tool change.
- Use CSS for turning, facing, and boring, NOT threading or drilling.
- When facing or turning small diameters, you will always reach the RPM limit at some diameter. This is easily calculated by rearranging our RPM formula like this:

$$D = Diameter = \frac{12 \times V}{p \times N}$$

Example of Cutting Speed – Diameter Limit

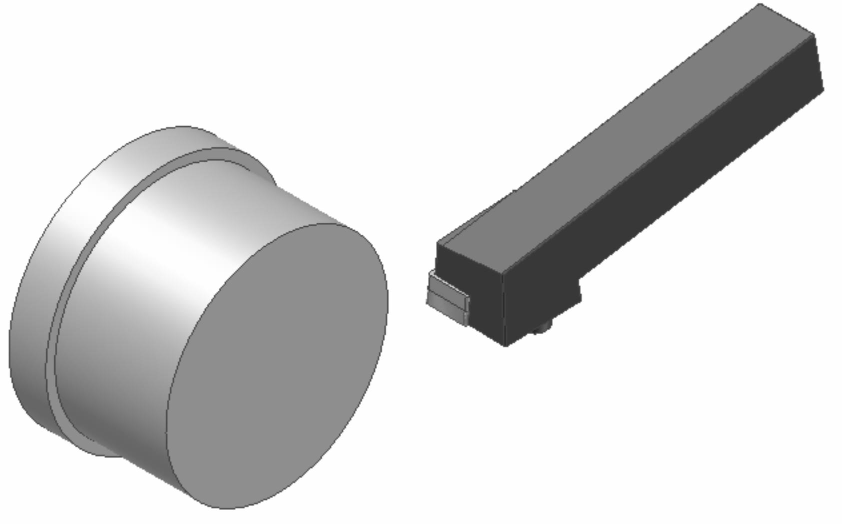
From our previous program, we used 600 fpm. At an RPM limit of 4000RPM and 600fpm cutting speed, we can calculate the diameter as:

$$D = Diameter = \frac{12 \times V}{p \times N} = \frac{12 \times 600}{p \times 4000} = 0.573''$$

So, any X value less than 0.573” means the cutting speed will be less than 600fpm with the corresponding poorer surface finish and shorter cutter life.

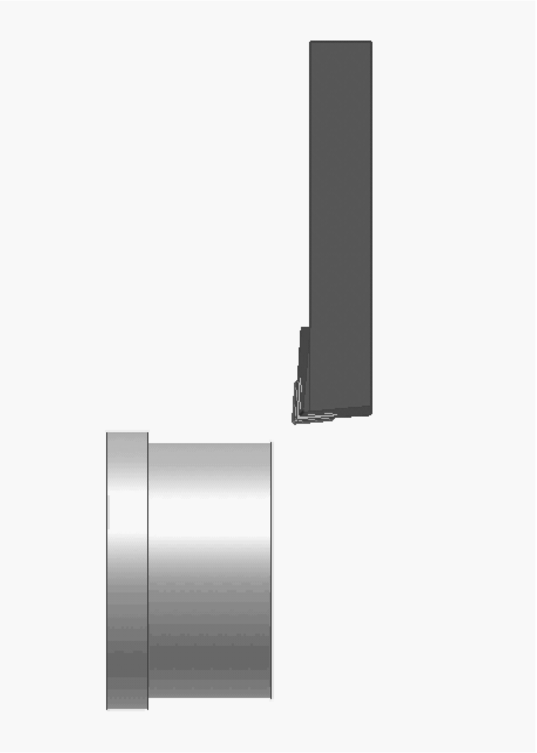
CNC Applications

Rectangular Cycles for Turning Centers



A Common Task

Select this link to
start the animation.



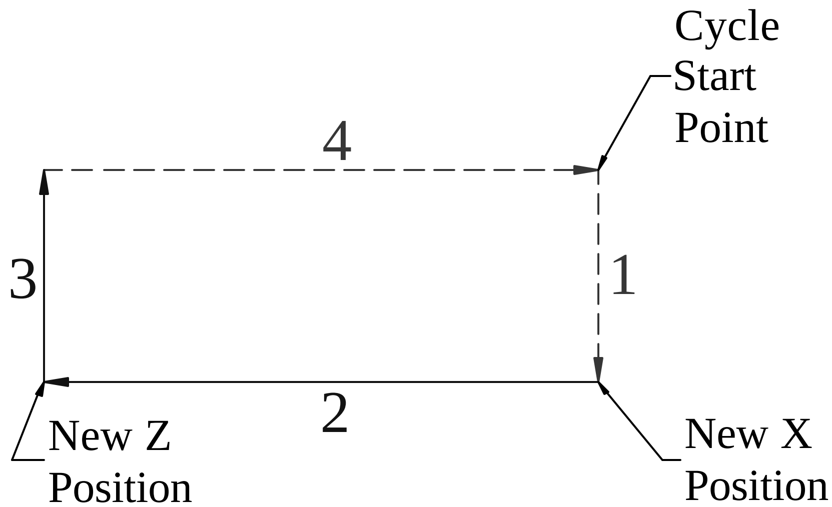
This is a common turning task – rapid to depth, feed to length, feed clear of the diameter, rapid back to the starting point.

This cut takes four blocks to program, a rapid, two feeds, and another rapid.

What is a Cycle?

- Normally, we only perform one positioning or cutting task in each block.
- However, tasks such as the four block turning sequence just provided are so common that control designers have incorporated CYCLES to reduce programming time.
- A cycle combines multiple moves into a single programmed block.

The Rectangular Turning Cycle



To use the rectangular turning cycle, you must first position the cutter at the Cycle Start Point. The cutter will also end up at the cycle start point at the end of the cycle. The program looks like this:

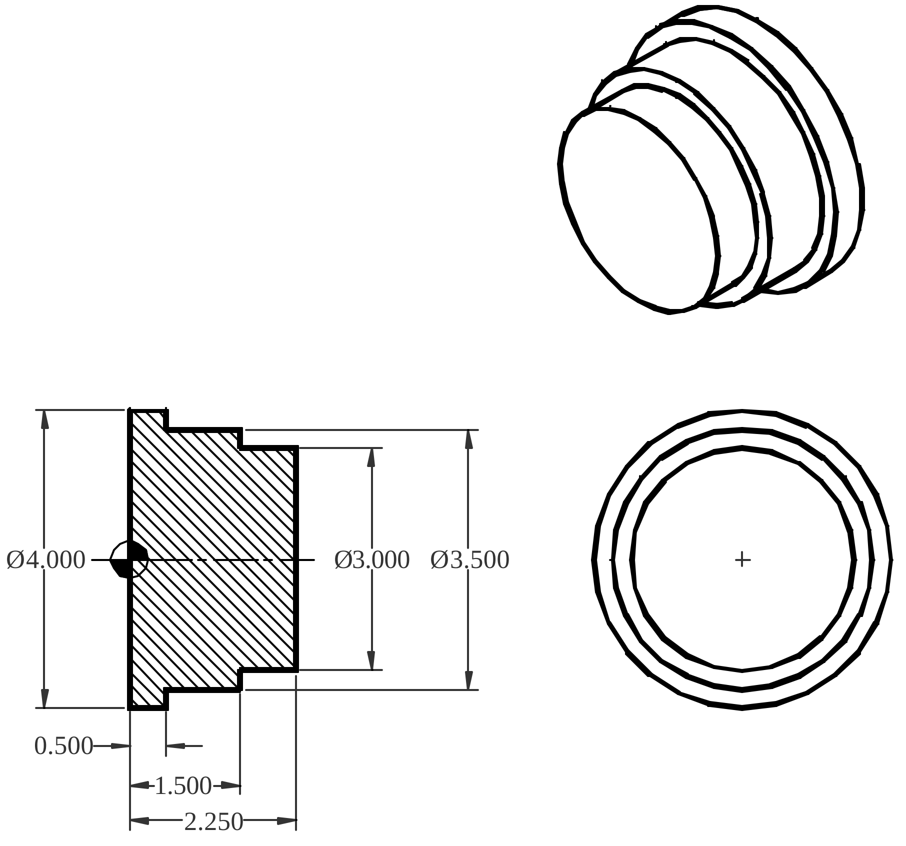
```
G90 Xnewx Znewz Fnewf
```

The cutter rapids to the newx diameter, feeds to the newz length, feeds to the starting X value, and then rapids back to the starting Z value. You get four blocks for just one programmed!

Additional G90 Notes

- The rectangular turning cycle (G90) is modal which is handy for repeat cuts.
- You can change newx and newz in succeeding cuts, and you can add a feed rate on any cut or just use the previous one.
- G90 only works in the direction shown. There are additional cycles for facing, boring, and turning towards the tailstock.
- What does G90 do on a machining center? (Hint: it has nothing to do with cycles.)

A G90 Turning Example



We will turn the 3.000” and 3.500” diameter steps on this part at 800fpm using the 80 degree C shaped insert in tool 2.

Follow Planning and Programming Steps (1-5)

1. Examine drawing
2. How will we hold the raw material – in a 3 jaw chuck.
3. Decide what cutters to use – given hard, coated carbide C shaped insert, and the cutting speed is also given (800fpm). We'll use constant cutting speed (css) and let the machine calculate and adjust the RPM based on the X position of the cutter.
4. Write down the exact sequence of operations:
   A. Rapid position the cutter in Z 0.25" away from the face.
   B. Rapid position the cutter 0.125" away from the part in X (radial).
   C. Take 0.125" radial cuts (0.25" from diameter) using the G90 turning cycle. Note: we are not taking finishing cuts in this example.
   D. Program end.
5. Convert the sequence of operations to a program:

Program Start

Turn the Steps

Program End

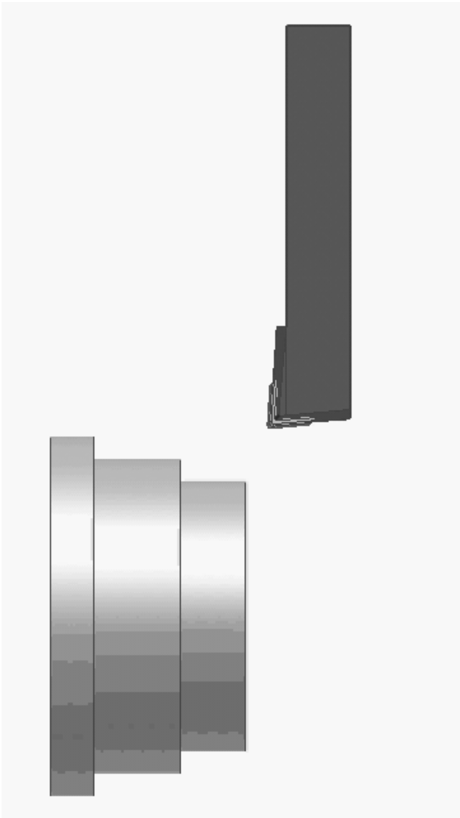
The Program

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0202	
G54	
G50 S4000	Cap the RPM
G96 S800 M3	Set the cutting speed to 800fpm, forward direction.
G0 Z2.5	Rapid 0.25 away from part in Z.
X4.25 M8	Rapid 0.125” radial distance from the part in X, coolant on.
G90 X3.75 Z0.5 F.012	First rectangular cycle cut removing 0.25” from diameter.
X3.5	Finish the larger step, G90 is still active.
X3.25 Z1.5	First cut on the second step, notice new Z value.
X3.0	Final cut on the second step.
M9	
M5	
G28 U0	Program End
G28 W0	
M30	
%	

The Animation

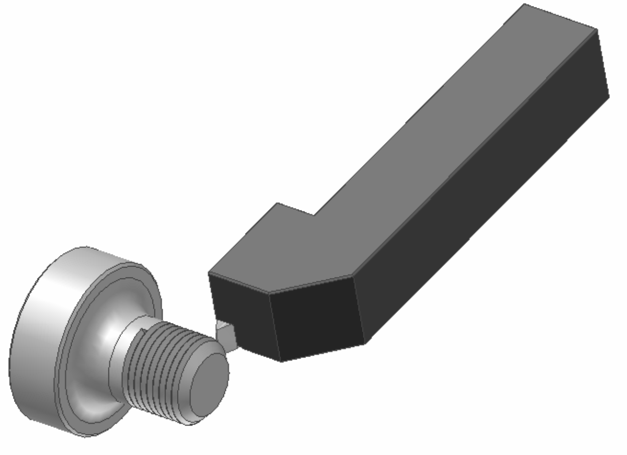
Select this link to
start the animation.

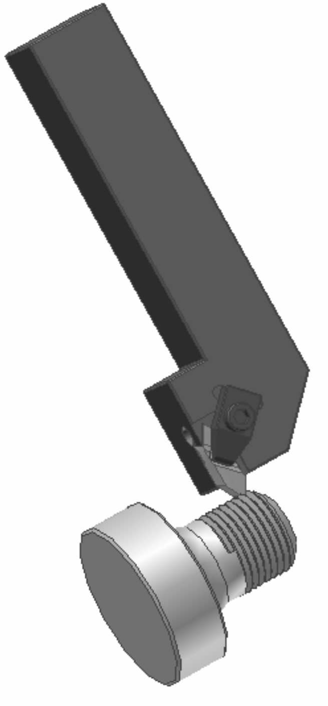
Remember, the animation does not show the difference between rapid and feed moves. When actually run on the machine, the rapid moves are much faster than the feed moves.



CNC Applications

Threading on Turning Centers





Conventional vs. CNC Threading

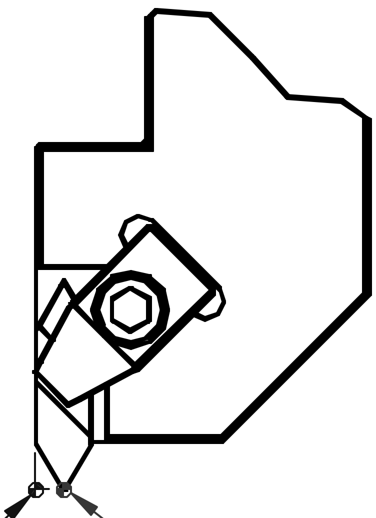
Conventional	CNC
Accuracy is dependant on the lead screw and gears.	Accuracy is dependant on the Z axis ballscrew and the electronics.
Manually synchronize multiple cuts with a threading dial	Electronically synchronize multiple cuts.
Cutting speed is limited by the operator’s ability to engage the half nuts.	Cutting speed is limited by how accurately the machine can synchronize the feed with the spindle RPM.
Takes about 10 minutes for an experienced operator to thread a ¾-16 UNF 2A 1” long.	Takes less than 1 minute for a good turning center to thread a ¾-16 UNF 2A 1” long.

Notes on CNC Threading

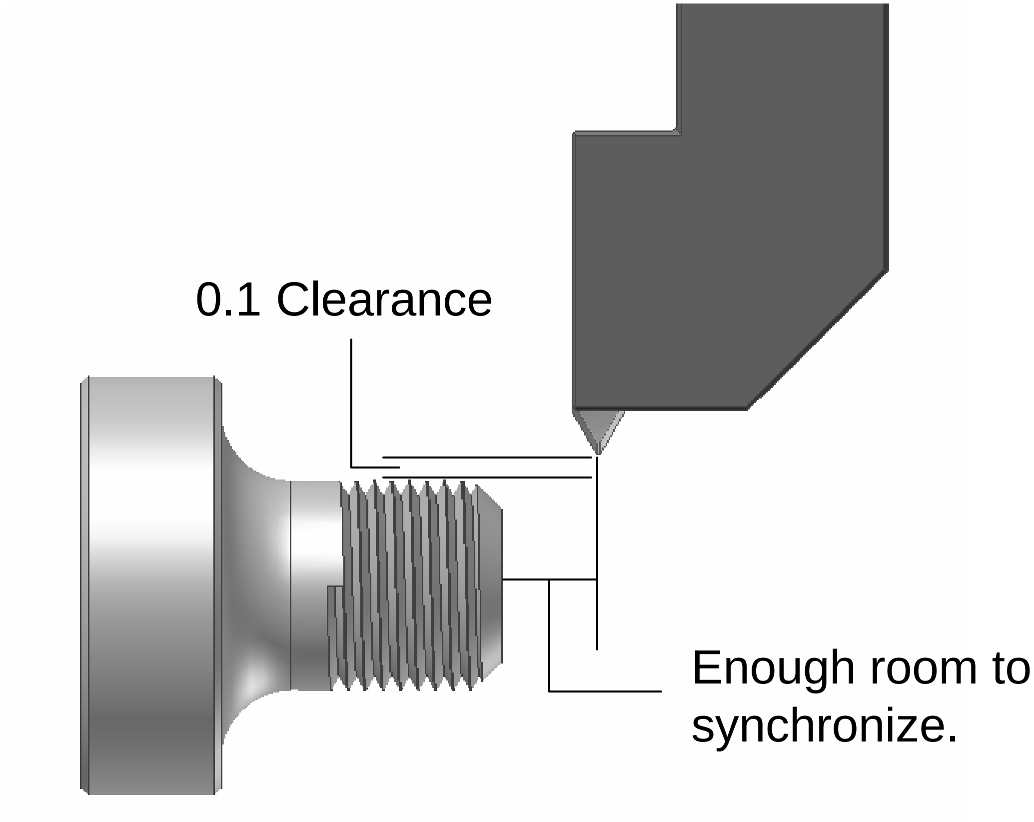
- Threading on a turning center is much faster than conventional threading because:
  - The machine can synchronize the feed and spindle RPM much faster than a person can engage the half nuts.
  - Faster synchronization means higher cutting speeds are used on CNC equipment resulting in faster production, better thread finishes, and more accurate threads.
  - The high rapid traverse rates re-position the cutter for subsequent cuts much more quickly than a person can.
- A CNC machine can cut any thread – English or Metric – without special equipment.

Threading Tool Offsets

You can set the tool offset for a threading tool at the tip or at the side. The X value is the same in either case, only the Z value differs. As noted below, setting at the side helps prevent running into shoulders but may not have enough threads, while setting at the tip gives the correct thread length with increased risk of hitting a shoulder. Be aware of the method being used.



Insuring Thread Accuracy

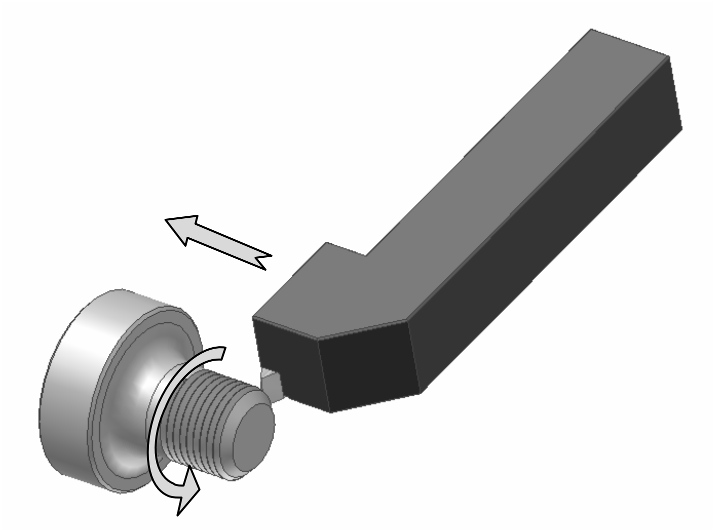


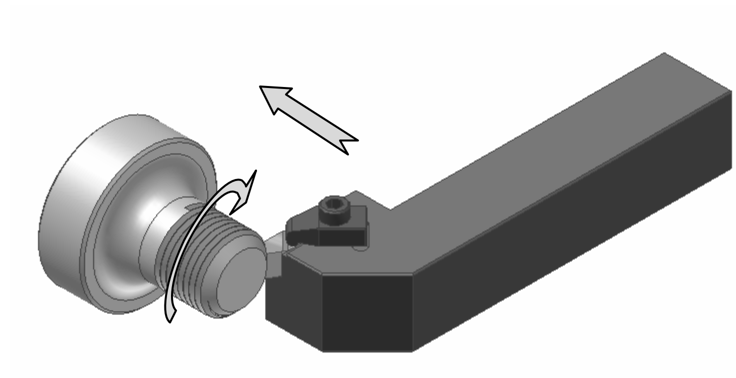
Be sure you have enough room to move around a live center or tailstock if one is being used!

While CNC machines are fast, they are not infallible. You must have enough clearance between the end of the part and the cutter's start point for the feed motors to synchronize with the spindle. Most machine manuals have a formula for this distance which depends on RPM and thread pitch.

If you cut a thread that has the correct pitch diameter but still won't fit a GO gage, increase this distance.

Right Hand or Left Hand?





Right Hand Thread – spindle is going forward (M3) and the cut is towards the headstock.

Left Hand Thread – spindle is going reverse (M4) and the cut is towards the headstock.

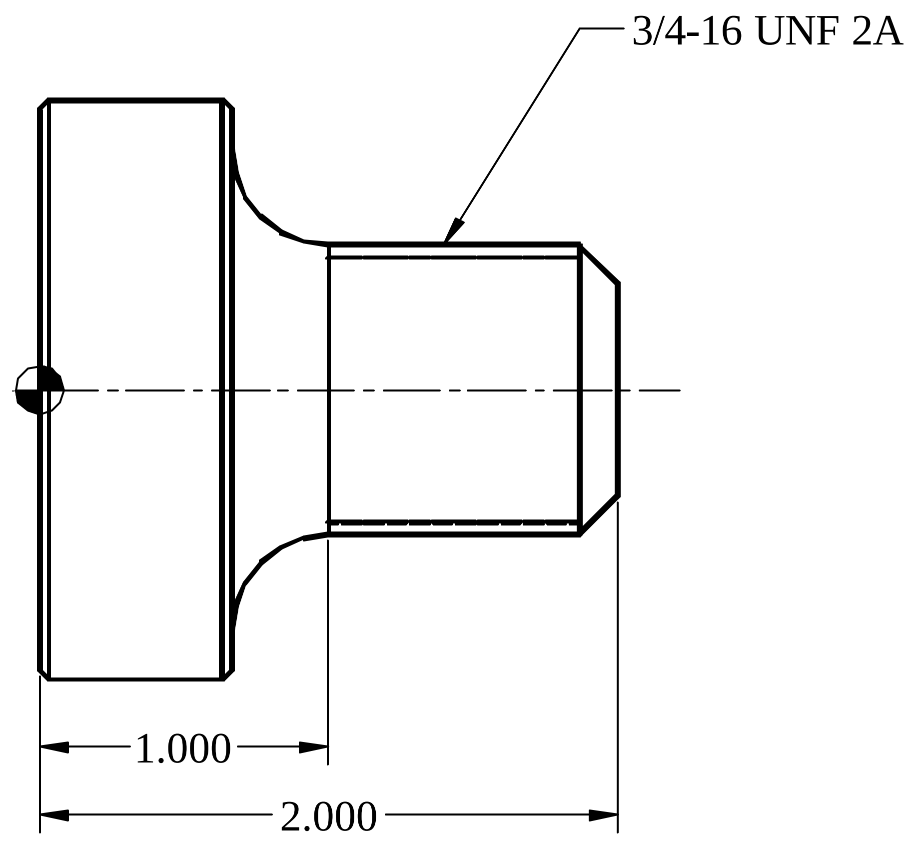
To complicate things further, we can reverse the spindle rotations, cutter hands, and cut directions shown above and end up with the same thing. However, the two pictures shown above are the most common methods of threading, so be sure you understand them.

G Codes for Threading

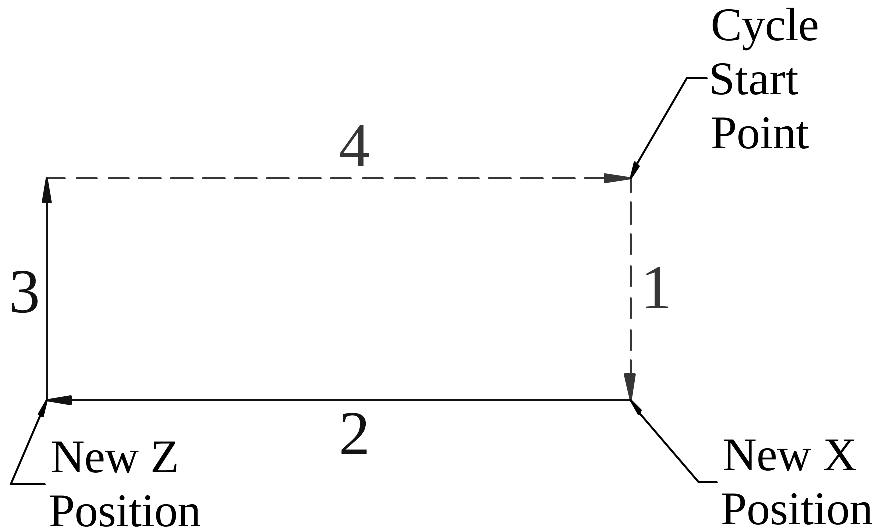
G Code	Application
G32	Requires four blocks per cut, mostly obsolete now.
G92	Works similarly to the G90 turning cycle with one block per cut required.
G76	The whole thread is cut with one block. This is the most common form of threading.

A Threading Example

We'll do this program twice – once with G92 and again with G76. In both cases, we'll assume that the profile is already turned, and we will do the threading at 400fpm.



The G92 Rectangular Threading Cycle



G92 works the same as G90 except for the synchronization between spindle and cutter to create the threads. Start the cutter at the Cycle Start Point. The cutter will also end up at the cycle start point at the end of the cycle. Each block looks like this:

```
G92 Xnewx Znewz Flead
```

Note that lead is actual pitch calculated as 1/tpi for single start threads. Also, many turning centers use E instead of F on threading cycles. Know your machine!

Follow Planning and Programming Steps (1-3)

1. Examine the drawing. We have to find some data from the *Machinery's Handbook* for a ¾-16 UNF 2A thread:
    - Major Diameter Range: 0.7391-0.7485”
    - Minor Diameter (Maximum): 0.674”
    - Lead = Pitch = 1/tpi = 1/16 = 0.0625”
2. How will we hold the raw material – in a collet chuck.
3. Decide what cutters to use – given a tough, coated carbide threading insert, and the cutting speed is also given (400fpm). We have to calculate the RPM since CSS should not be used when threading:

$$N = RPM = \frac{12 \times V}{p \times D} = \frac{12 \times 400}{p \times 0.75} = 2037$$

Follow Planning and Programming Steps (4-5)

4. Write down the exact sequence of operations:
   A. Rapid position the cutter in Z 0.25” away from the face.
   B. Rapid position the cutter 0.1” away from the part in X (radial).
   C. Based on the Machinery’s Handbook data, we’ll assume the blank is 0.745” diameter to start, and we’ll take 6 passes at the following X values:

0.725	0.705
0.690	0.680
0.677	0.674

   D. Program end.
5. Convert the sequence of operations to a program:

Program Start

Make the Threading Passes

Program End

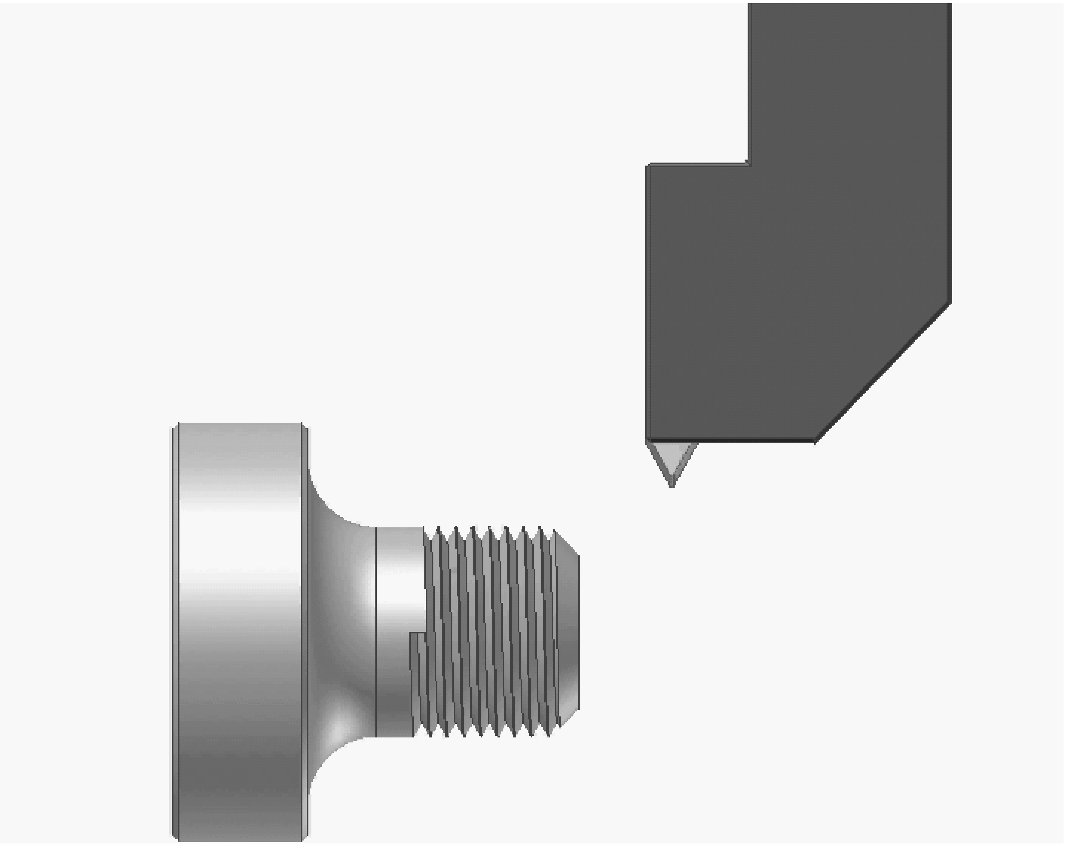
Note: on our machine, the threading tool offset is taken from the side, not the point, of the cutter, so our threads will be somewhat short which we can adjust for in the program if we need to.

The Program with G92

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0505	Load the threading tool
G54	
G97 S2037 M3	Set the RPM, forward direction.
G0 Z2.25	Rapid 0.25 away from part in Z.
X0.945 M8	Rapid 0.1” radial distance from the part in X, coolant on.
G92 X0.720 Z1.0 F0.0625	First threading cycle cut removing 0.020” from diameter.
X0.705	Second threading cut, G92 is still active.
X0.690	
X0.680	
X0.677	
X0.674	Final threading cut, just a light pass.
M9	
M5	
G28 U0	Program End
G28 W0	
M30	
%	

Threading Animation

Select this link to
start the animation.



Note: the animation does not show the cutter moving to the start point or to home after machining the thread. It only shows the G92 cycle blocks.

Again, this is just an animation. The machine would cut the thread much faster than the animation shows.

G76 Threading Cycle

Cutting threads is so common, the CNC designers have created the G76 cycle to cut the entire thread in one pass. The format looks like this:

`G76 X``rootx` **`Z`**`endz` **`I`**`taper` **`K`**`height` **`D`**`pass1` **`F`**`lead` **`A`**`angle`

Where: rootx = minor diameter of the thread (required)

endz = the ending Z value of the thread (required)

taper = amount of taper when cutting a tapered thread (optional)

height = radial height of the thread (required)

pass1 = depth of the first pass (Note, most machines do not allow a decimal point on D, so an integer must be used.) (required)

lead = pitch for a single start thread which is 1/tpi (required)

angle = angle to enter the thread (optional)

General Comments about G76

- If you leave I off, the cycle produces a straight thread which is most common.
- If you leave A off, the cutter feeds straight in (see the next two slides for a more detailed description of A).
- The cycle works for both ID and OD threads based on the cycle start point and the values of X and Z.
- The cycle automatically decides how many passes to take depending on the value of K and D. Each pass is smaller than the previous pass.
- Some machines have more control over the number of passes and the depth of the final pass. Know your machine!

The K and D Codes

K is the height of the thread and is easily calculated with the following formula.

$$K = Thread\ Height = \frac{Major\ Diameter - Minor\ Diameter}{2}$$

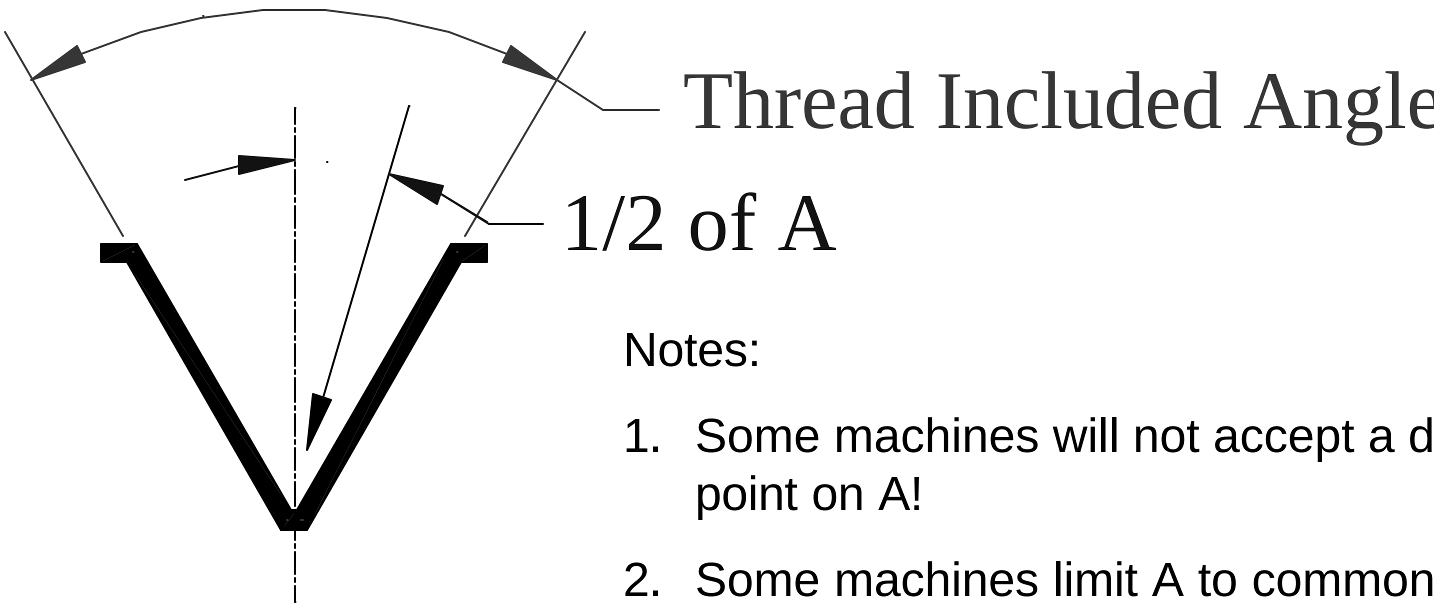
When looking in the Machinery's Handbook for thread specifications, remember that the major diameter for an external thread is given as a range, so use the high side of the range. For our ¾-16 UNF 2A:

$$K = Thread\ Height = \frac{0.748 - 0.674}{2} = 0.037$$

D is the depth of the first pass, usually in integer form. If you want a 0.012" deep first pass, set D to 0120. Remember the resolution of most machines is 0.0001", so 0120=0.0120".

Deciphering the A Code

As shown in the diagram below, a thread has an included angle. The most common angles are 60º for both metric and inch V threads and 29º for ACME threads. By changing the value of A, we can change the infeed angle of the threading cutter. The infeed angle is always ½ the value of A specified in the G76 cycle.

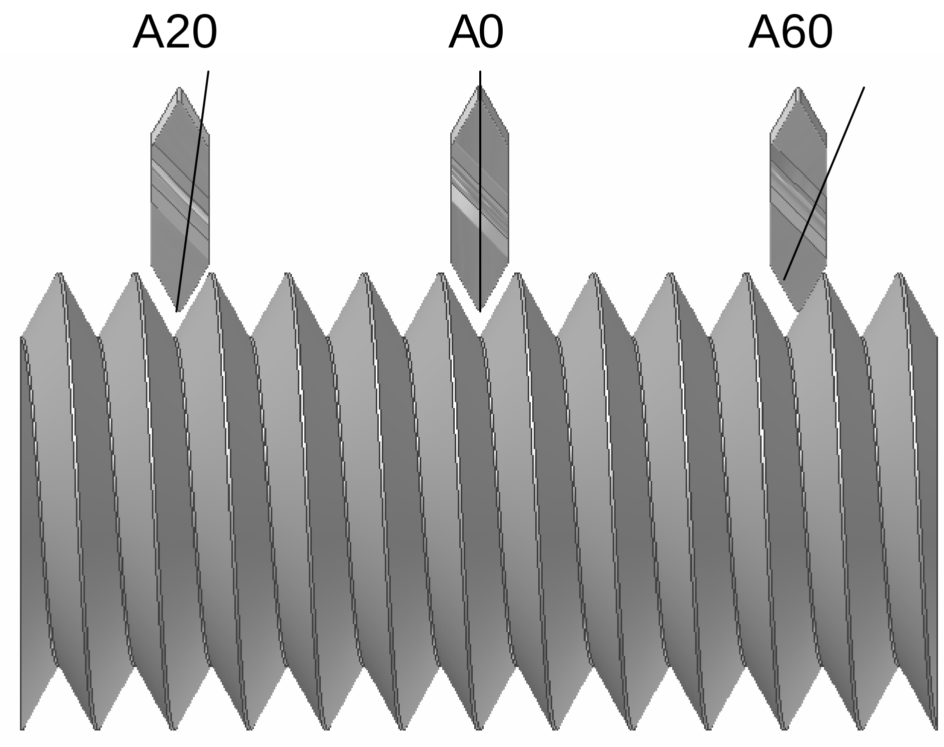


Notes:

1. Some machines will not accept a decimal point on A!
2. Some machines limit A to common thread angles, others allow any value.

Threading Feed Angle

A20 – the machine actually feeds at 10º. Most cutting takes place on the leading flank, but some takes place on the trailing flank. A good compromise since it is fairly easy on the cutter, leaves a good finish, and tends to minimize chatter.



A60 – the machine actually feeds at 30º, or down the trailing flank. All cutting takes place on the insert's leading edge, which is easiest on the cutter. The trailing flank usually has a poor finish. This is how most conventional (manual) threading is done.

A0 – the default. The cutter feeds straight in, and the insert cuts equally on both flanks. This is hard on the cutter, but both flanks usually have a good finish.

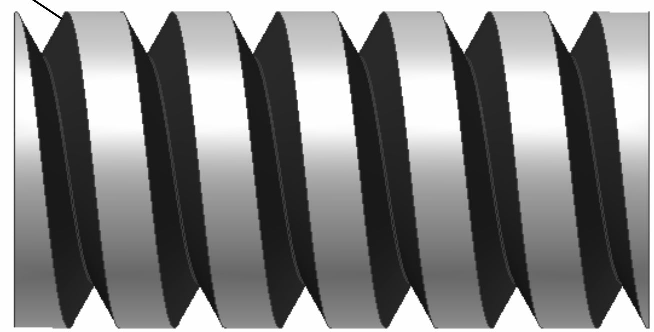
The Program with G76

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0505	Load the threading tool
G54	
G97 S2037 M3	Set the RPM, forward direction.
G0 Z2.25	Rapid 0.25 away from part in Z.
X0.945 M8	Rapid 0.1” radial distance from the part.
G76 X0.674 Z1.0 D0120 K0.037 A20 F0.0625	Cuts the entire thread in one block!
M9	
M5	Program End
G28 U0	
G28 W0	
M30	
%	

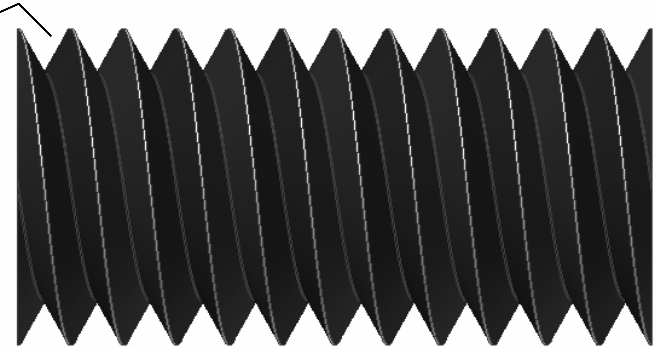
Cutting Multiple Lead Threads

For single start threads (the most common), the lead is equal to the pitch. Or, in threading terms, the amount of advancement for one turn is equal to the distance between the threads.

Occasionally, we have to cut multiple start threads where the lead is an even multiple of the pitch. For example, to cut a ¾-16 double lead thread, we would cut a ¾-8 thread half-way deep like this:



Then, we move the start point of the threading cycle over by the pitch (0.0625") and cut another thread in between those we just cut like this:

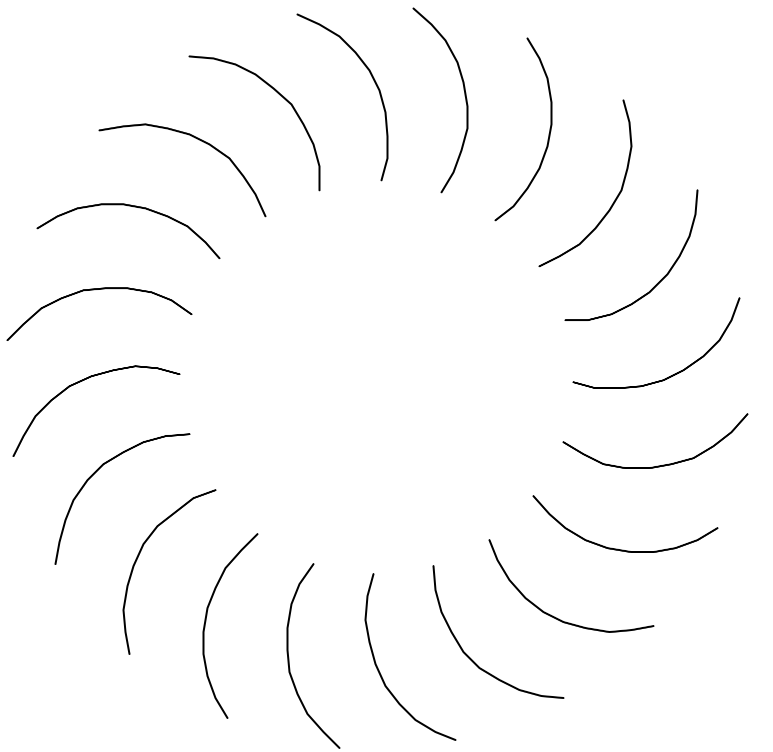


Cutting a Double Lead Thread with G76

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0505	Load the threading tool
G54	
G97 S2037 M3	Set the RPM, forward direction.
G0 Z2.25	Rapid 0.25 away from part in Z.
X0.945 M8	Rapid 0.1” radial distance from the part.
G76 X0.674 Z1.0 D0120 K0.037 A20 F0.125	Cuts the first thread, note the lead.
G0 Z2.3125	Re-position the starting point.
G76 X0.674 Z1.0 D0120 K0.037 A20 F0.125	Cut the second thread.
M9	
M5	
G28 U0	Program End
G28 W0	
M30	
%	

CNC Applications

Programming Arcs



Why Program Arcs?

- Many components have radius features which require machining.
- Arc programming on turning centers eliminates the need for form tools and results in a better finish.
- For machining centers, we can easily cut arcs which would otherwise require a complicated setup on a rotary table.
- For machining centers, internal radii such as the corner of pockets always machine better with an arc move rather than depending on the cutter to leave the radius.
- We have much more flexibility in choosing cutters on both machining and turning centers.

Arc Overview

To program an arc, you must know the coordinates of the following three points:

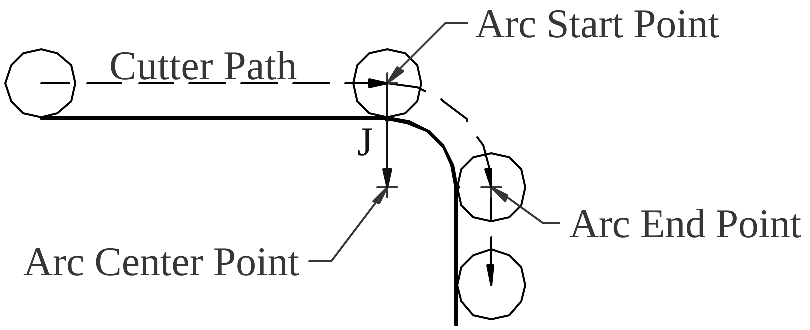
1. Arc Start Point
2. Arc End Point
3. Arc Center Point

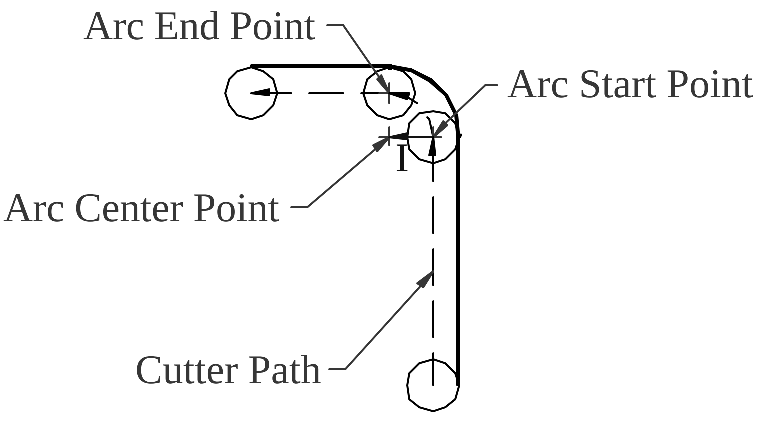
Notes:

On machining centers, you are programming the center of the cutter, so you must account for the radius of the cutter.

The cutter must be tangent to the arc at the start point and at the end point.

G02 Clock Wise Arc





G03 Counter Clock Wise Arc

General Format for Arc Blocks on Machining Centers

G2 Xendx Yendy Ivectorx Jvectory Fnewf

G3 Xendx Yendy Ivectorx Jvectory Fnewf

Where:

endx,endy are the coordinates of the Arc End Point.

vectorx,vectory are the X and Y distances from the Arc Start Point to the Arc Center Point.

newf is a new feed rate, if desired. If Fnewf is left off, the last active feedrate will be used (F is modal).

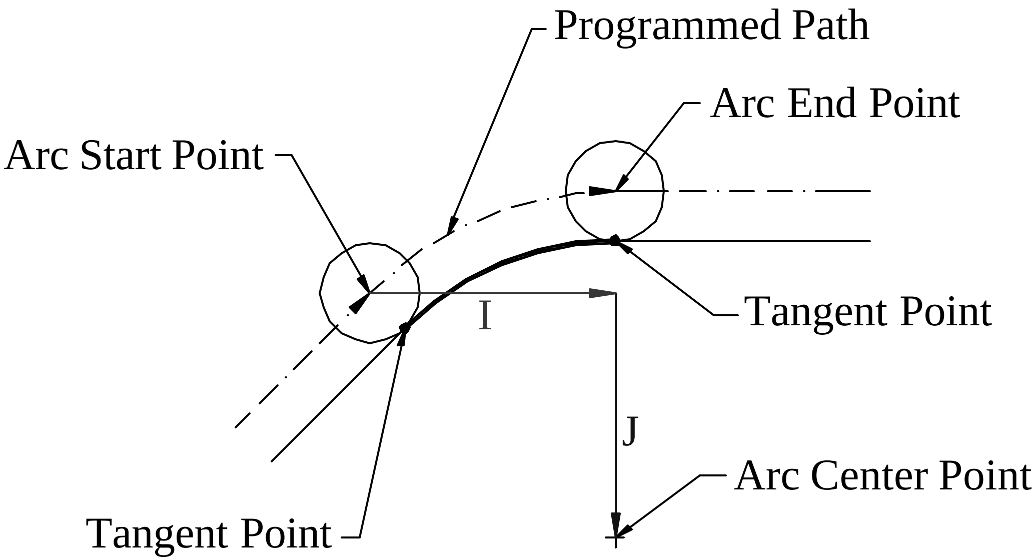
Refer to the picture on the previous page for definitions.

I and J for Machining Centers

Many people have trouble understanding I and J when they are really quite simple. I and J are signed X,Y directions from the Arc Start Point to the Arc Center point.

The illustration shows an arc of <90 degrees which has both I and J values. 0, 90, 180, and 270 degree arcs always have either I or J as zero.

Note for this example that I is a positive number while J is a negative number.



I and J (continued)

Mathematically, you can calculate I and J as:

$I = X_{ArcCenterPoint} - X_{ArcStartPoint}$ $\qquad$ $J = Y_{ArcCenterPoint} - Y_{ArcStartPoint}$

You can describe I and J as:

I=Distance from the Arc Start Point to the Arc Center point in X

J=Distance from the Arc Start Point to the Arc Center point in Y

Notice the difference between the mathematical definition and the written description of I and J. You can use either method to find I and J, but be sure you get the sign correct!

An Example With Numbers

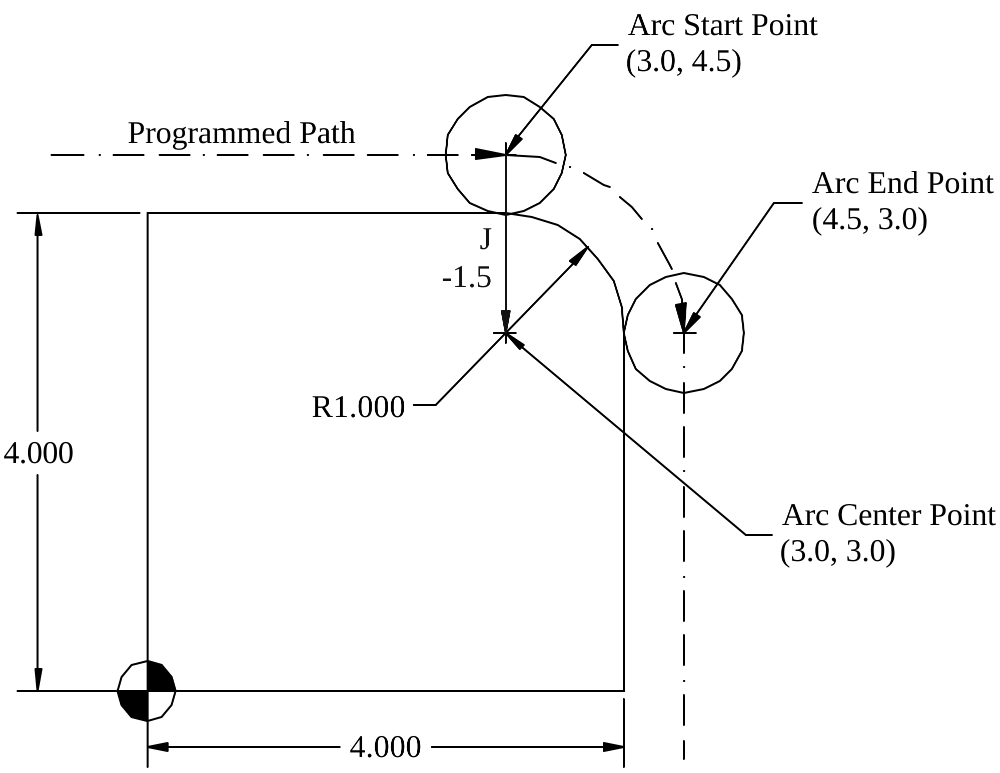
We'll mill the programmed path with a 1” diameter (0.5” radius) cutter. After finding the coordinates of the three points as shown, we calculate I and J as:

I = 3.0 – 3.0 = 0

J = 3.0 – 4.5 = -1.5

And the program segment would look like this:

```
G0 X-0.75 Y4.5
G1 Z-0.25
X3
G2 X4.5 Y3.0 J-1.5
G1 Y-0.75
```



Note that G2 and G3 are modal. A common mistake is to forget a G1 when a linear move follows an arc as in this example.

General Format for Arc Blocks on Turning Centers

G2 Xendx Zendz Ivectorx Kvectorz Fnewf
G3 Xendx Zendz Ivectorx Kvectorz Fnewf
Where:

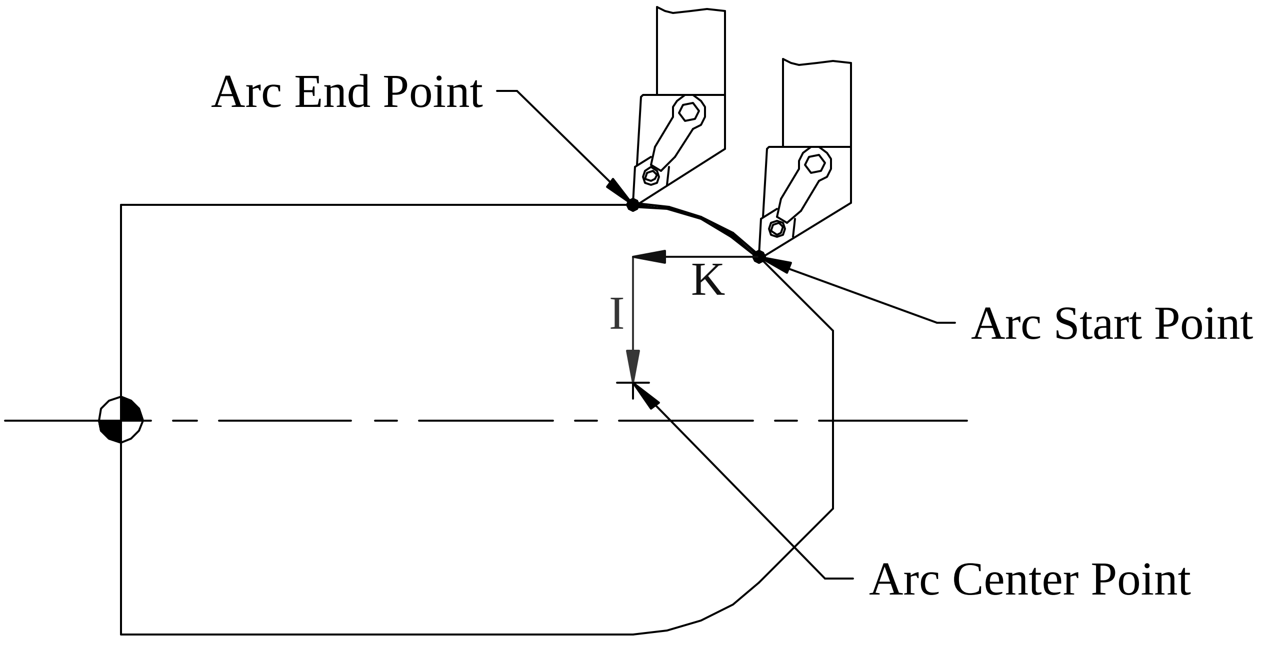
endx,endz are the coordinates of the Arc End Point.
vectorx,vectorz are the X and z distances from the Arc Start Point to the Arc Center Point.

newf is a new feed rate, if desired. If Fnewf is left off, the last active feedrate will be used (F is modal).

Note the only difference from machining centers is Y and J are replaced with Z and K.

I and K for Turning Centers

Note that the values for I are RADIAL even though we program X as diameter!



For turning centers, calculate I and K like this:

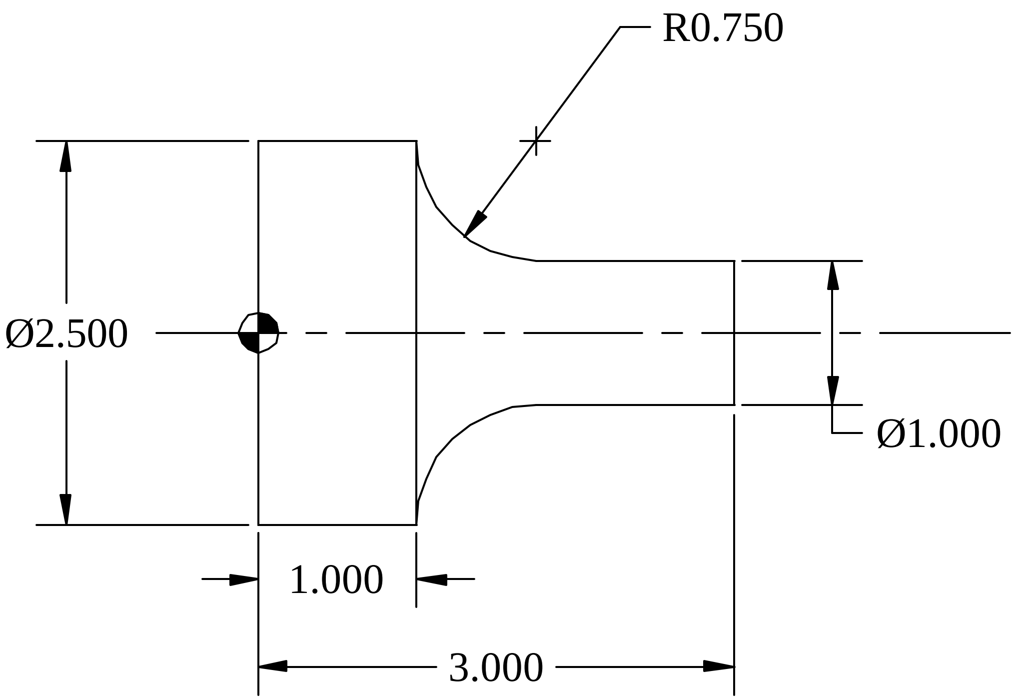
$I = (X_{ArcCenterPoint} - X_{ArcStartPoint})/2$ and $K = Z_{ArcCenterPoint} - Z_{ArcStartPoint}$

Special Notes for Arcs on Turning Centers

- Machining center arc programming must allow for the radius of the cutter, turning center arc programming generally does not.
- The insert nose radius for a turning cutter does cause some inaccuracy in the arc formation which we will address later in the course.
- For now, ignore the insert nose radius and just remember it is a problem we will solve shortly.

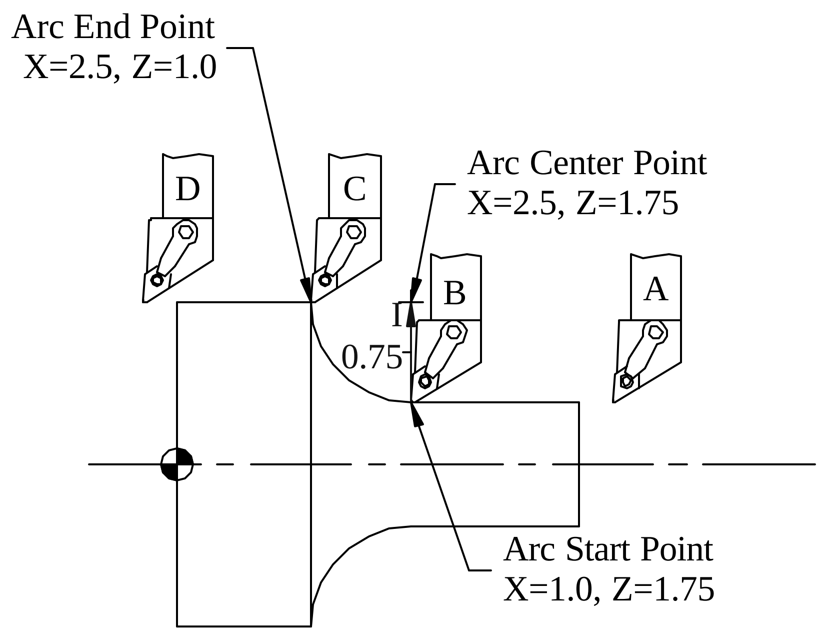
A Turning Center Example

We’ll take a finish pass across the 1” diameter, the 0.750” radius, and the 2.5” diameter using a 55 degree (shape D) carbide insert cutter.



How the Machine Moves

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0303	
G54	
S2800 M3	
G0 Z3.25	A. Rapid to position in
X1.0 M8	Z and X, coolant
G1 Z1.75 F.006	B. Feed to Arc Start
G2 X2.5 Z1.0 I0.75	C. Form arc
G1 Z-0.25	D. Feed clear in Z
X2.75	and X
M9	
M5	
G28 U0	
G28 W0	
M30	
%	

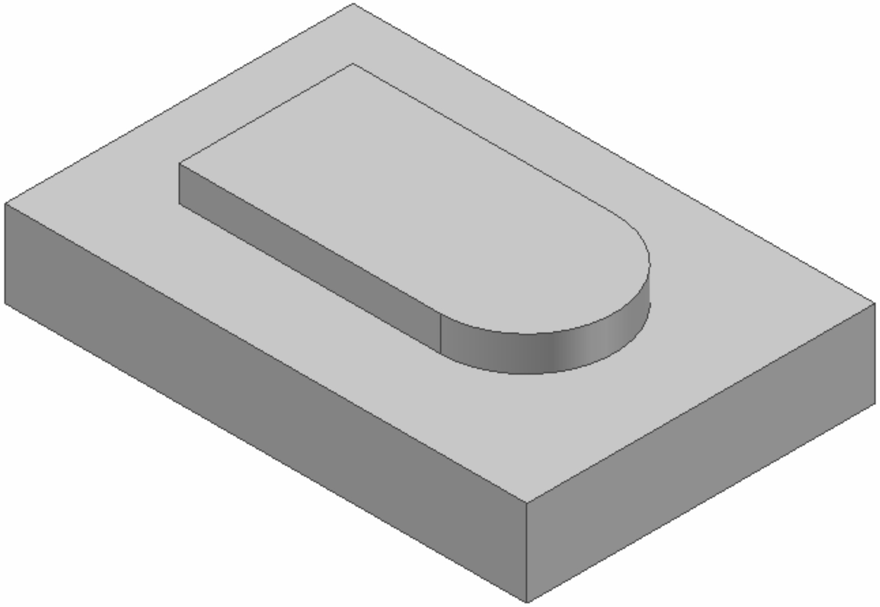


I = (2.5 – 1.0)/2 = 0.75

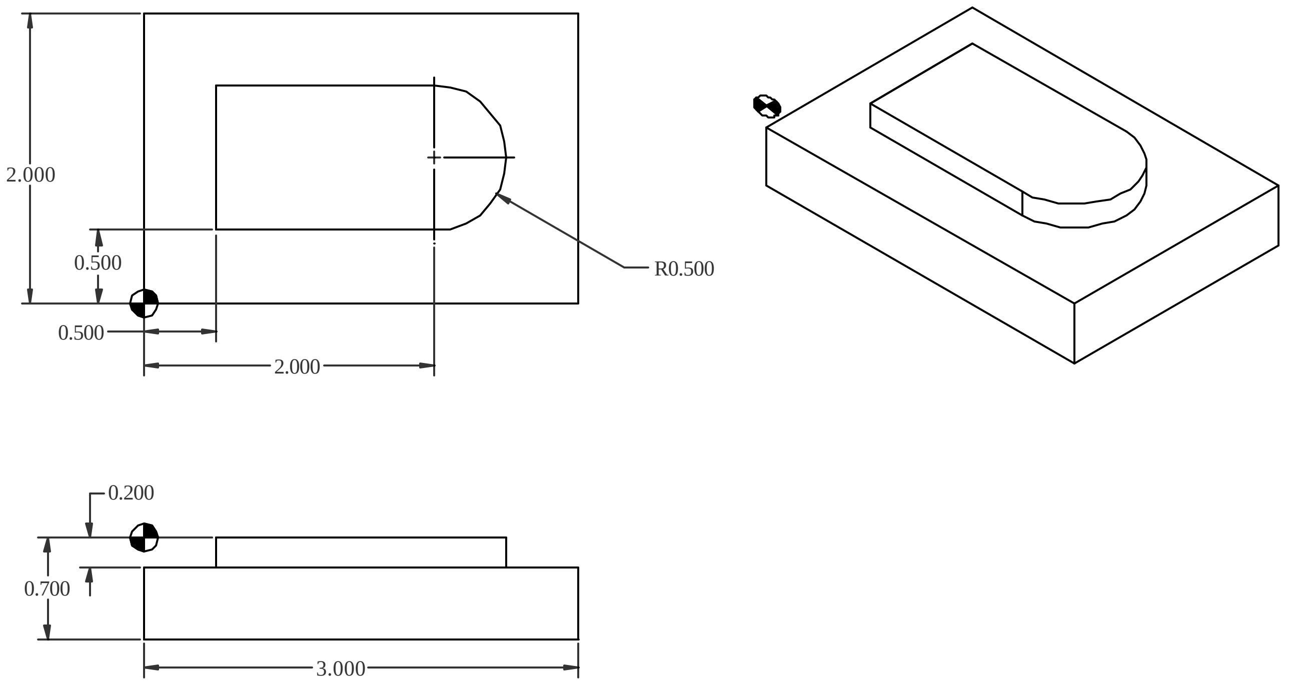
K = 1.75 – 1.75 = 0

CNC Applications

Machining Center Example #2



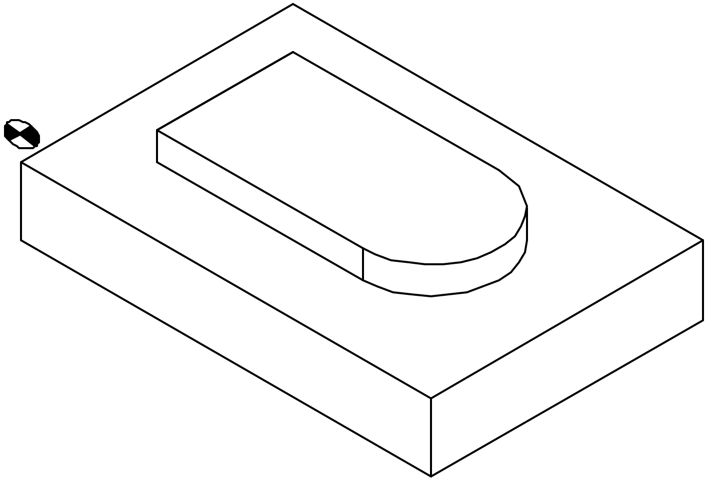
Problem Statement



Machine this part from ¾”x2” 6061 CD aluminum saw cut to 3 1/8” length using a ¾” 2 flute HSS endmill (tool 1). Accuracy requirements on the profile are high, so a rough cut 0.010” away followed by a finish cut are required on the profile only.

Planning and Programming (1)

1. Examine the part drawing thoroughly and get a rough idea of how you want to proceed.
   A. Pick the WCS in the lower left corner of the part on the finished upper surface before machining the profile:
   B. Machine one end with the part against a stop.
   C. Program stop, flip the part, and machine the 3” length.
   D. Machine 0.050” off the top of the part leaving the final 0.700” thickness.
   E. Rough and finish the profile.
   F. Remove remaining “tails” in the corners.



Planning and Programming (2-3)

2. How will we hold the part? In a 6” vise up on 1/8” wide parallels that hold the part only 3/8” into the vise jaws. A stop on the right positions the part.
3. Decide what cutters to use – given a ¾” diameter 2 flute HSS endmill. From the Machinery’s Handbook, we note that this endmill has 1 5/16” of useable flute length.

Planning and Programming (3 continued)

3. For the endmill, we find from Table 10 that V=600fpm and from Table 15a f_t=0.004ipt:

$$N = RPM = \frac{12 \times V}{p \times D} = \frac{12 \times 600}{p \times 0.75} = 3056$$

$$f_m = f_t\ n_t\ N = 0.004 \text{ x } 2 \text{ x } 3056 = 24 \text{ ipm}$$

Planning and Programming (4)

4. Write down the exact sequence of operations:

   A. Rapid position cutter 1/16” to the left and clear in Y
   B. Feed to depth, face left end of the part, rapid up
   C. Move home in Y,Z and then flip the part
   D. Repeat A and B except for final X position
   E. Face the extra 0.050” off the top of the part
   F. Rough machine the profile leaving 0.010”.
   G. Finish machine the profile.
   H. Machine off excess material left in the corners nearest the radius.
   I. Program end.

Planning and Programming (5)

5. Convert the sequence of operations to a program:

Program Start
Machine Left End
Flip Part
Machine to Length
Machine Thickness
Rough Profile
Finish Profile
Machine Corners
Program End

Don’t Redo Work

- Notice that the machine-to-length and machine-to-thickness operations required for this part are exactly the same as in Machining Center Example 1.

- We will simply copy the first program and add additional blocks to create this program.

- This is easily done by doing “Save As” in Notepad or other editor and then adding to the new file.

The Length and Thickness Program

Program Codes (1)

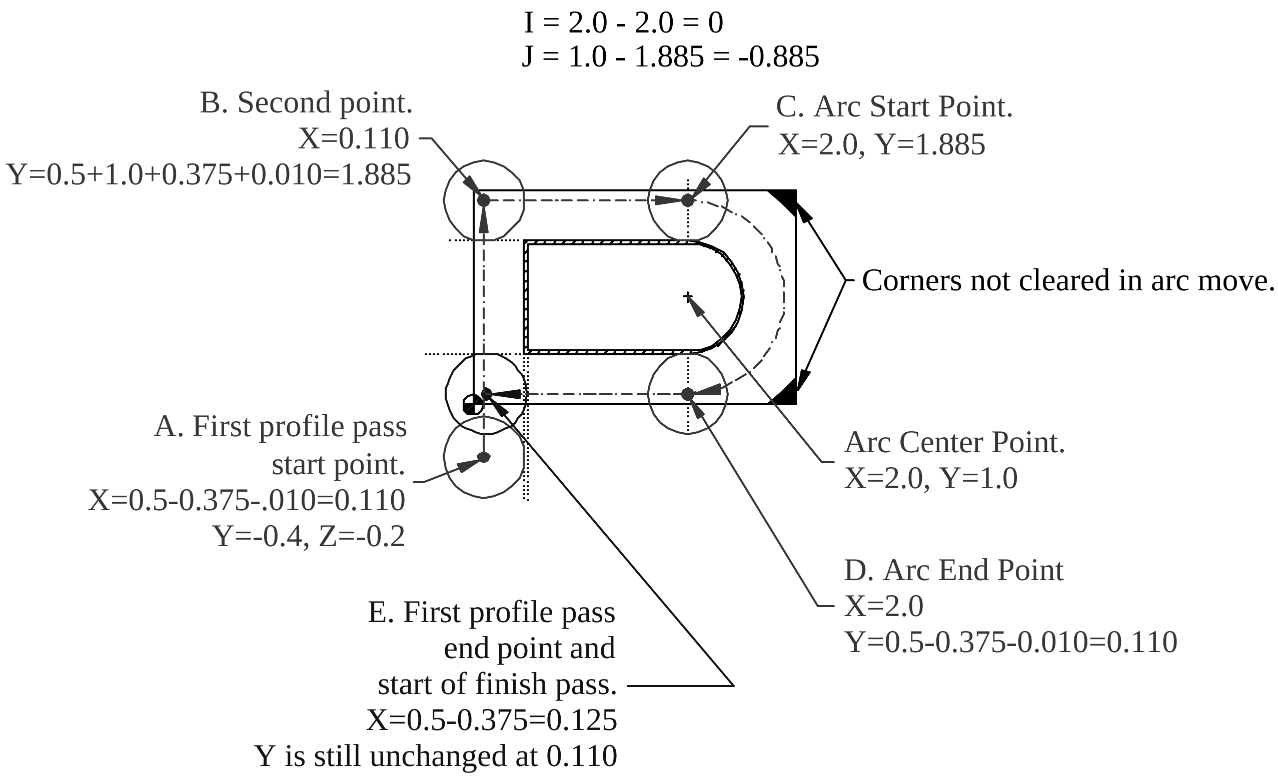
```
%
O999
G20 G40 G49 G80 G99
G91 G28 Z0
G90
T1M6
S3056 M3
G0 G90 G54 X-0.437 Y-0.4
G43 H1 Z0.1 M8
G1 Z-0.8 F24.
Y2.1
G0 Z0.1
M9
M5
G91 G28 Z0
G28 Y0
G90
M0
```

Program Codes (2)

```
M3
G0 X-0.375 Y-0.4
Z0.1 M8
G1 Z-0.8
Y2.1
G0 Z0
Y1.75
G1 X3.4
G0 Y1.125
G1 X-0.4
G0 Y0.5
G1 X3.4
G0 Y0
G1 X-0.4
```

Add the profile and corner machining here.

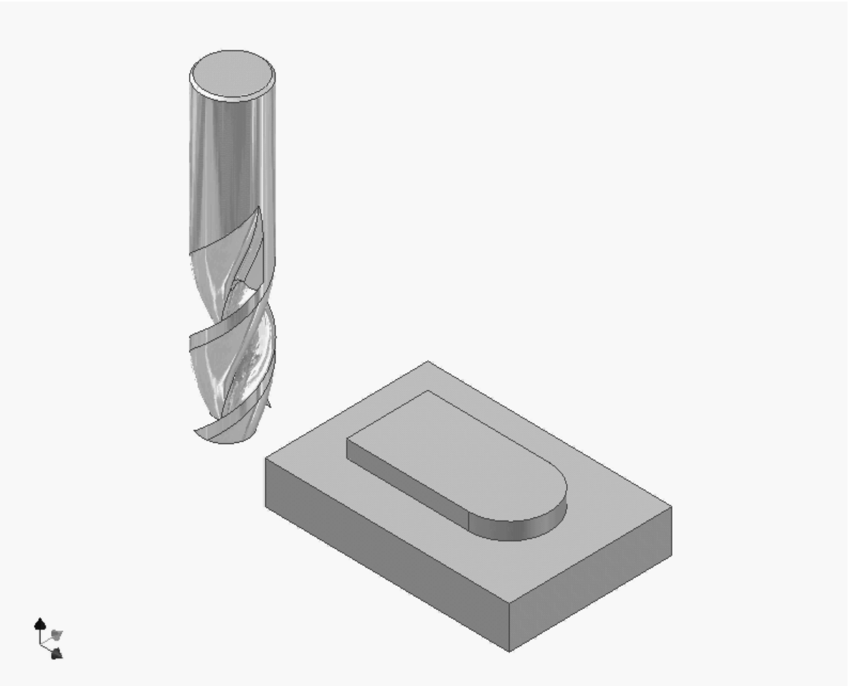
Overview of the First Profiling Pass



We will rapid position to point A and then follow points B-E to perform the first rough pass. Note the calculations for I and J, and that material is left in the two rightmost corners that will have to be removed later.

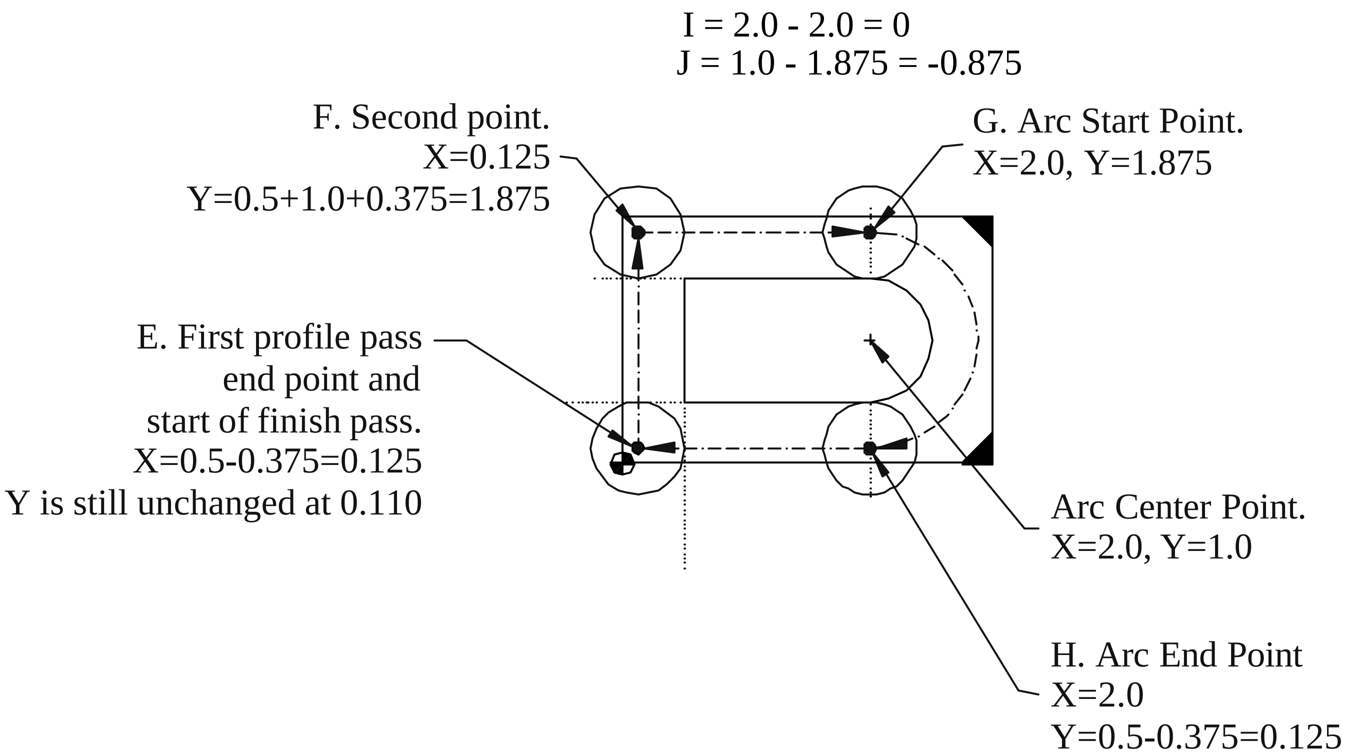
The Blocks for the First Profile Pass

Program Codes	Action
G0 Y-0.4 X0.110 Z-0.2 G1 Y1.885 X2.0 G2 Y0.110 J-0.885 G1 X0.125	. Position at point A. . Feed to point B. Feed to point C. Machine arc to point D. Feed to point E.



Select this link to start the animation.

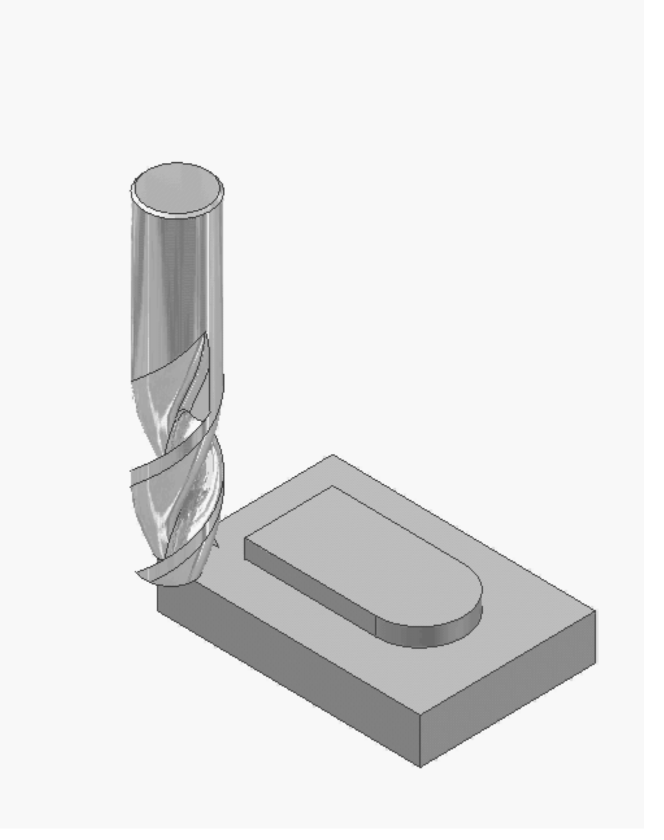
Overview of the Finish Profiling Pass



We will continue feeding without pause after the roughing pass to points F-H. Then, we will feed completely off the part to the left in preparation for machining the excess corner material. Notice that J has changed from the rough pass.

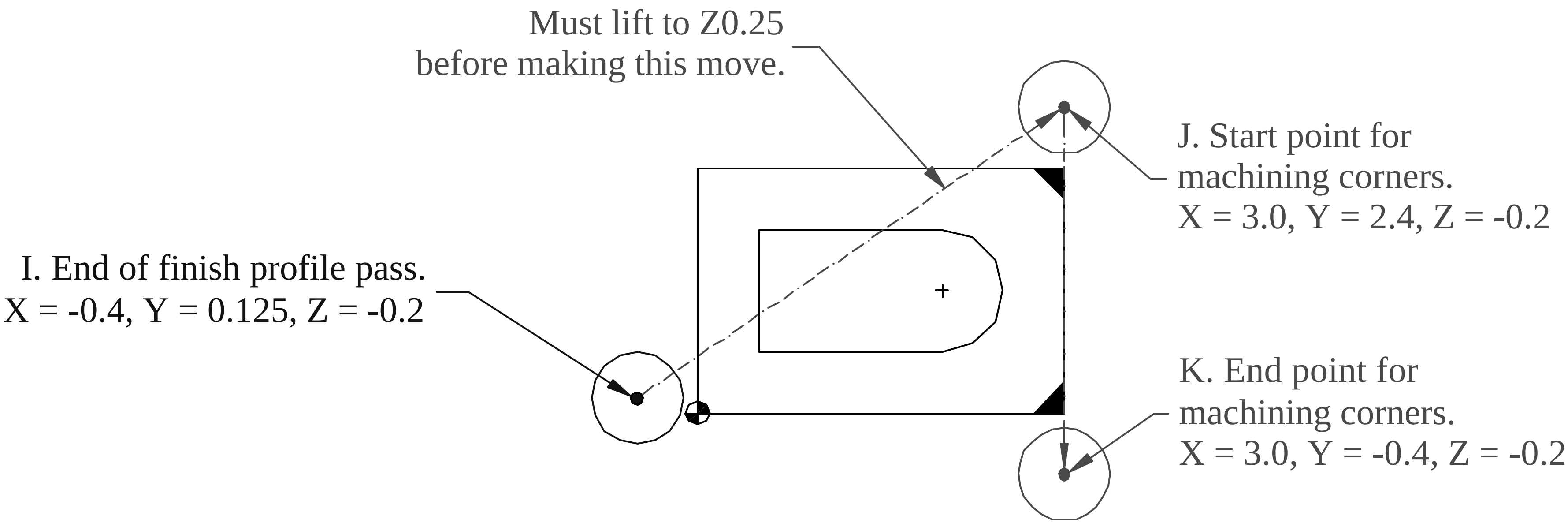
The Blocks for the Finish Profile Pass

Program Codes	Action
Y1.875 X 2.0 G2 Y0.125 J-0.875 G1 X-0.4	Feed to point F. Feed to point G. Arc to point H. Feed off the part.



Select this link to start the animation.

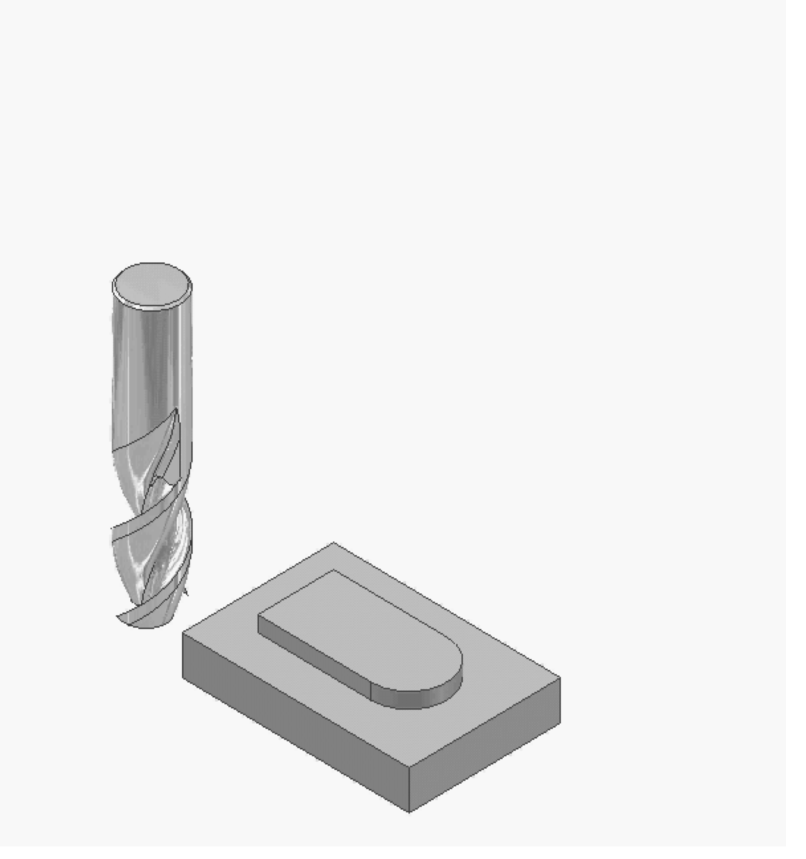
Overview of Machining Corners



After finishing the second profile pass, we'll rapid up to Z0.25 to clear the part and then move over to point J. We will then go back down to Z-0.2 and feed across the corners. The program ends by lifting up and then going home in Z and Y.

Machine the Corners

Program Codes	Action
G0 Z0.25 X3.0 Y2.4 Z-0.2 G1 Y-0.4 G0 Z0.25 M9 M5 G49 G91 G28 Z0 G28 Y0 G90 M30 %	Lift above the part. Rapid to point I. Go back down to depth. Feed to point J. Lift above the part. Program end . . . . .



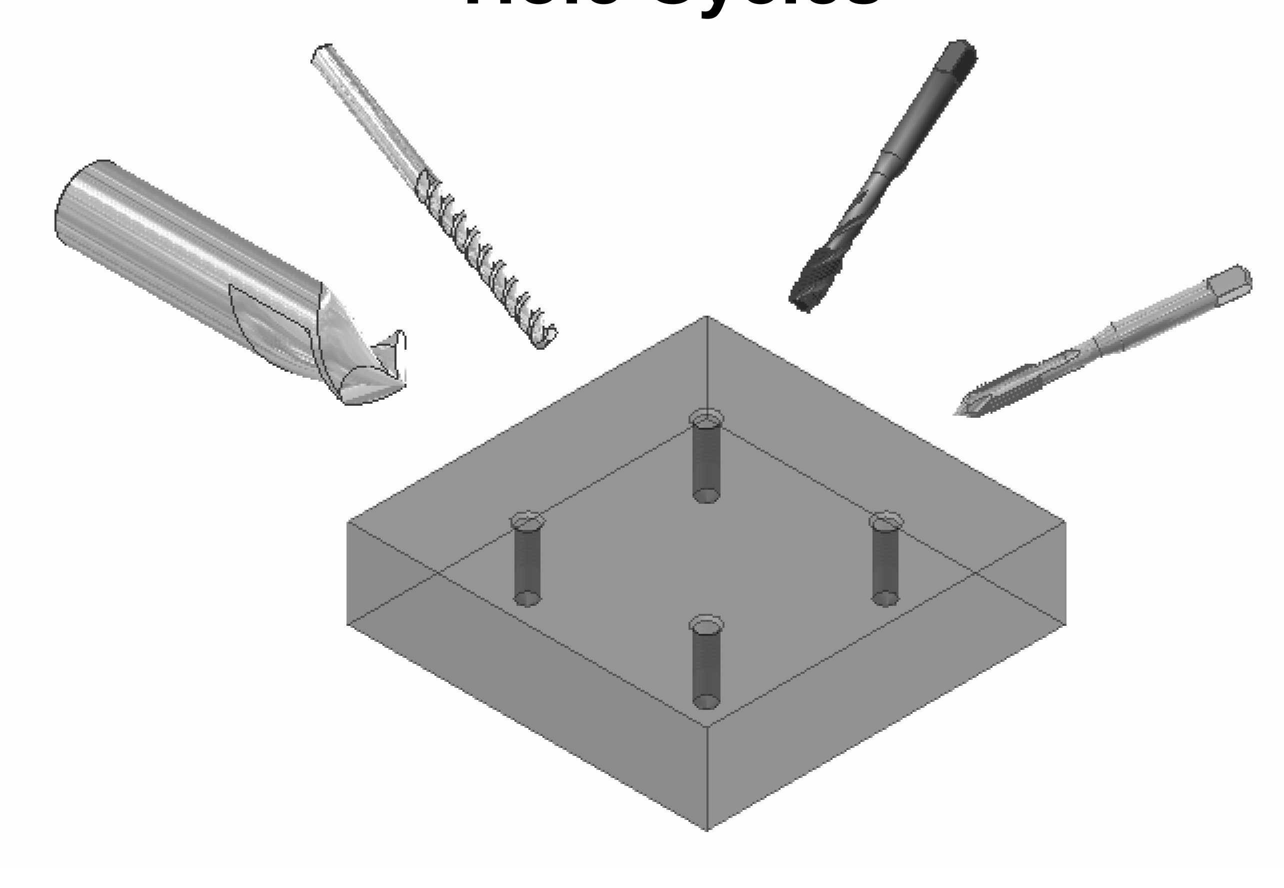
Select this link to start the animation.

The Final Program

Program Codes		
%	M3	G0 Z0.25
O999	G0 X-0.375 Y-0.4	X3.0 Y2.4
G20 G40 G49 G80 G99	Z0.1 M8	Z-0.2
G91 G28 Z0	G1 Z-0.8	G1 Y-0.4
G90	Y2.1	G0 Z0.25
T1M6	G0 Z0	M9
S3056 M3	Y1.75	M5
G0 G90 G54 X-0.437 Y-0.4	G1 X3.4	G49
G43 H1 Z0.1 M8	G0 Y1.125	G91 G28 Z0
G1 Z-0.8 F24.	G1 X-0.4	G28 Y0
Y2.1	G0 Y0.5	G90
G0 Z0.1	G1 X3.4	M30
M9	G0 Y0	%
M5	G1 X-0.4	
G91 G28 Z0	G0 Y-0.4	
G28 Y0	X0.110	
G90	Z-0.2	
M0	G1 Y1.885	
	X2.0	
	G2 Y0.110 J-0.885	
	G1 X0.125	

CNC Applications

Hole Cycles



Why so Many Hole Cycles?

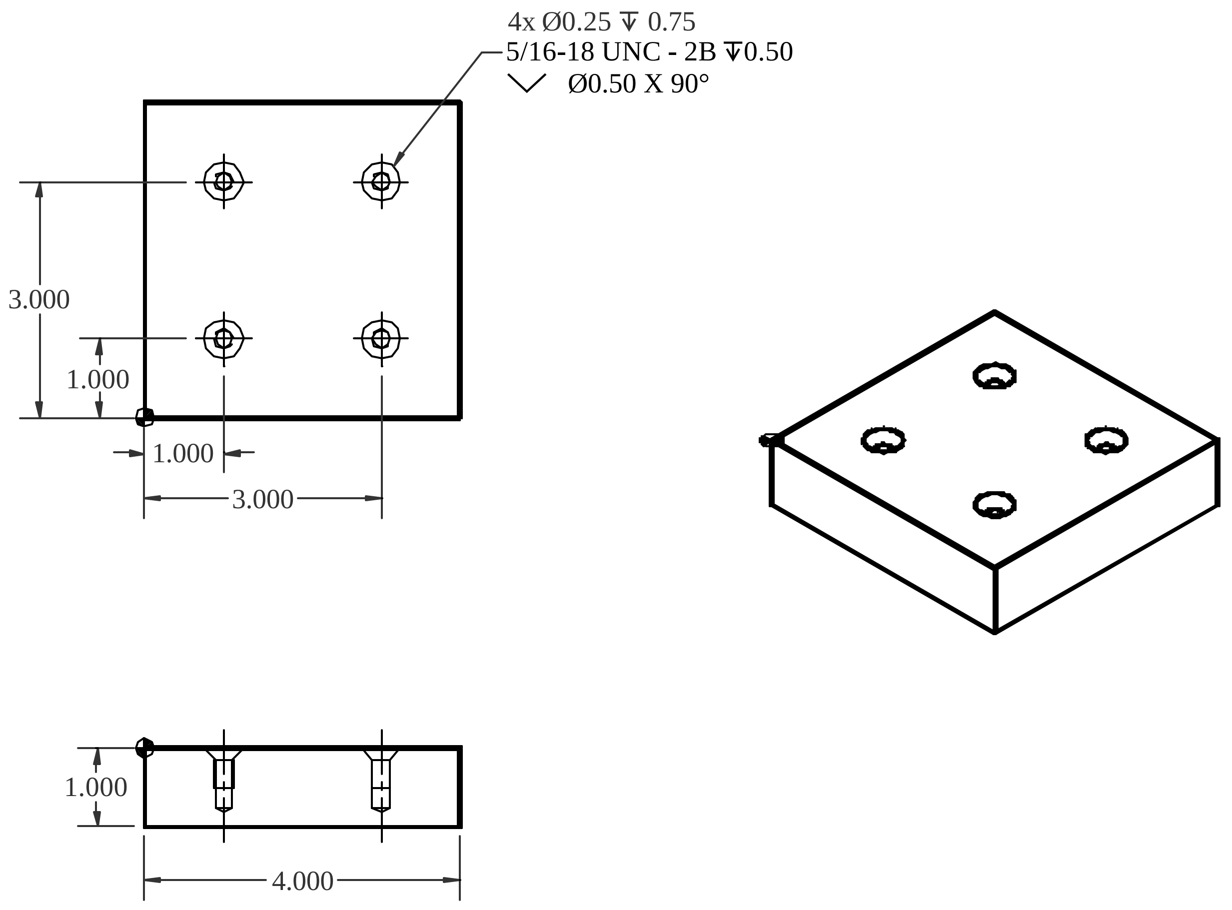
- Creating holes is the most common machining operation since nearly all machined parts have at least one hole.
- Machining centers have many hole cycles including drilling, deep hole drilling, peck drilling, tapping, boring, etc.
- Turning centers usually have fewer hole cycles than machining centers, but they still have generally drilling, peck drilling, and tapping.
- Hole cycles for machining and turning centers are usually very similar.

Codes for Hole Cycles

G Code	Application
G81	Simple drilling – feeds to depth and rapids out of the hole.
G83	Peck drilling – feeds in a specified distance, rapids out to clear chips, rapids back in, and repeats.
G84	Right hand tapping – feeds to depth at correct pitch, automatically reverses, and feeds out.
G80	Cancels any hole cycle.

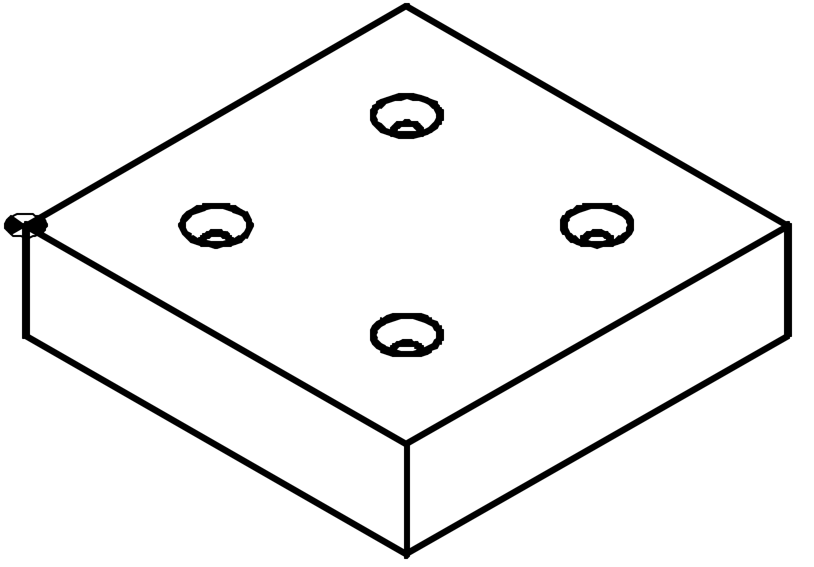
A Simple Example

We will learn the basics of hole cycles by programming a machining center to create the holes in this part. We will start with spot drilling, continue with peck drilling, and finish with tapping.



Planning and Programming (1)

1. Examine the part drawing thoroughly and get a rough idea of how you want to proceed.
   A. Pick the WCS in the lower left corner of the part on the upper surface:
   B. Assume length and thickness are already finished.
   C. Spot drill all holes deep enough for the countersink.
   D. Drill all the holes with a ¼” drill.
   E. Tap all the holes with a spiral flute tap since the holes are blind.



Planning and Programming (2-3)

2. How will we hold the part? In a 6” vise up on 1/8” wide parallels. These thin parallels will not interfere with the holes, even if they went through the part. A stop on the right positions the part.
3. Decide what cutters to use – a 5/8” diameter spot drill (T12), a jobber’s length ¼” drill (T19), and a 5/16-18 spiral flute drill (T20).

Spot drill RPM = 12x600 / (p x0.625) = 3667 RPM

¼” drill RPM = 12x600 / (p x0.25) = 9167 RPM, use 7000RPM

Tap RPM = choose 1500 RPM, machine will not tap at full RPM

Planning and Programming (4-5)

4. Write down the exact sequence of operations:
   A. Rapid cutters 0.1” above the part over the lower, left hole.
   B. Perform the spot drilling, then do the other three holes.
   C. Tool change, repeat A-B for the ¼” drill.
   D. Tool change, repeat A-B for the 5/16-18 tap.
   E. Program end.
5. Convert the sequence of operations to a program:

Program Start

Spot Drill Holes

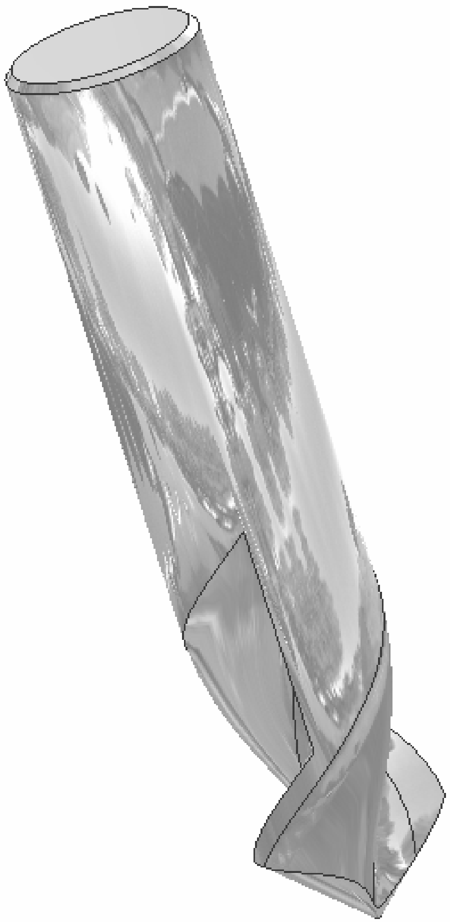
Peck Drill Holes

Tap Holes

Program End

Spot Drilling

When drilling holes on a machining center, a spot drill locates the hole prior to actual drilling. On conventional machines, centerdrills are generally used for the same function, but the spot drill has three advantages over the centerdrill:

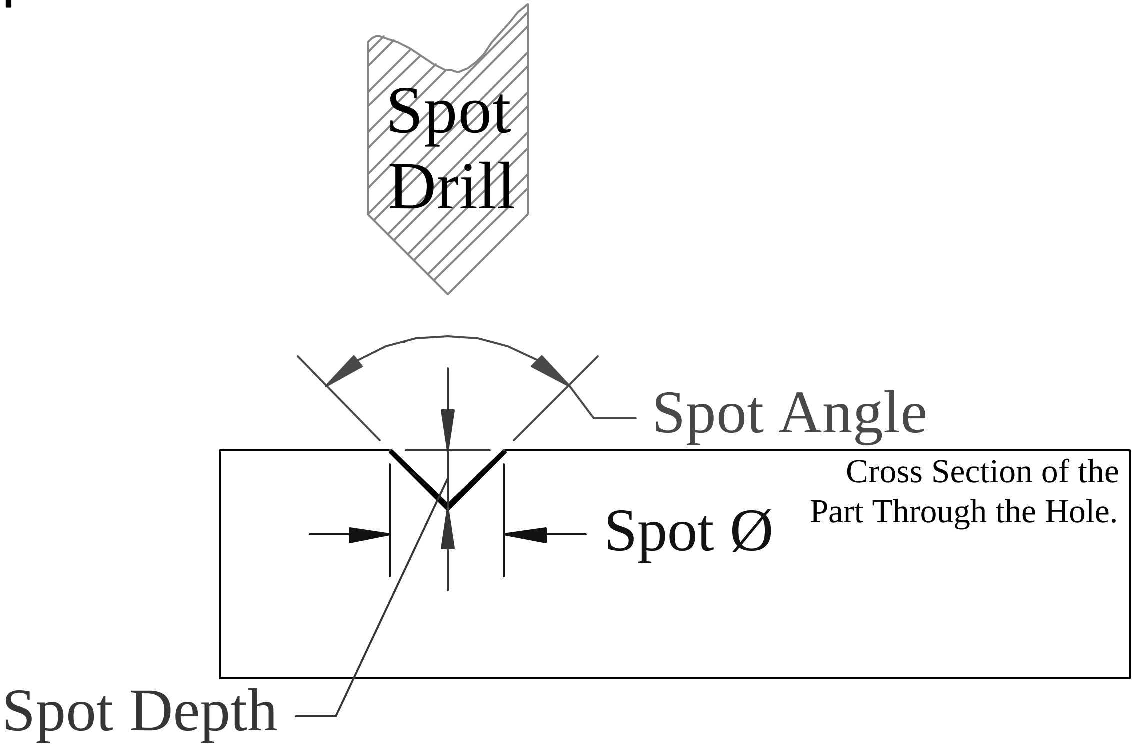


1. The spot drill does not have a small diameter point to break off and ruin the part.
2. By programming the spot depth carefully, we can generally eliminate countersinking.
3. Spot drills are commonly available with a 90º point angle which generates a 45º chamfer, and other angles are available if desired.

The primary application of a centerdrill is to provide a location for a live center point when turning. People generally use them to spot holes, but spot drills work better.

Spot Drill Givens

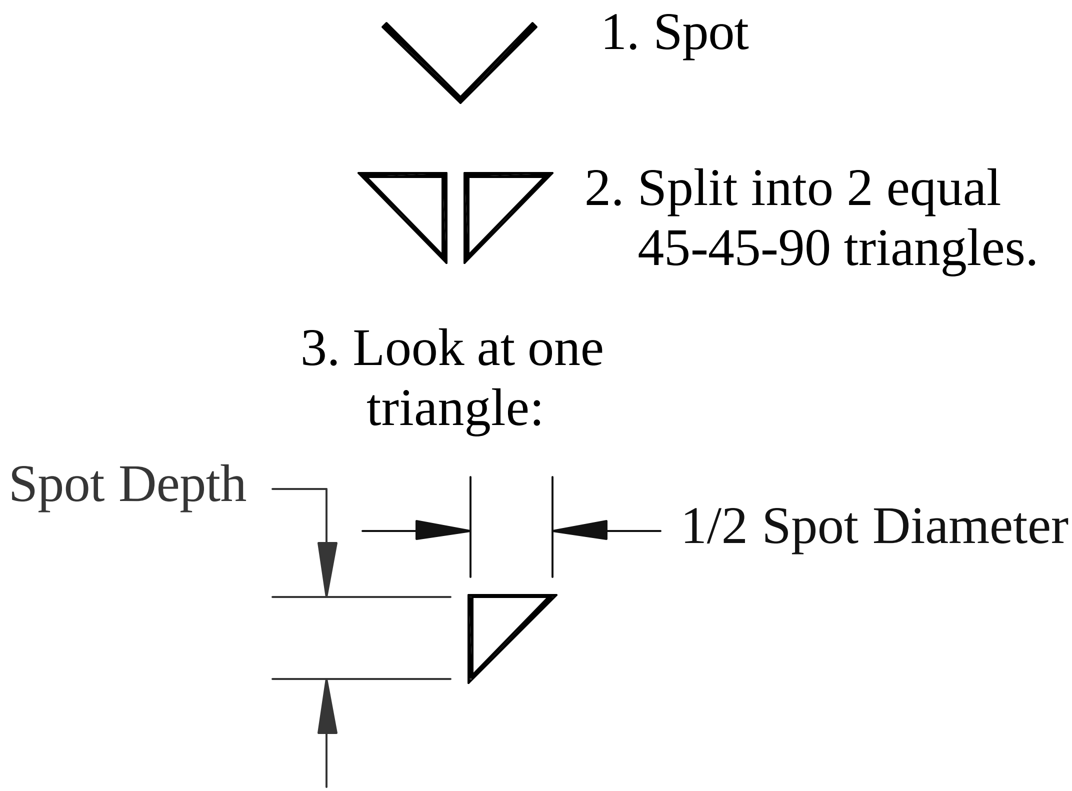
- When programming a spot drill, we know the spot diameter and the spot angle, but we have to program the spot depth.



Calculating Spot Drill Depth

Calculating spot depth with a 90º spot drill is quite easy as shown in these three steps. As you can see, spot depth is simply ½ of spot diameter.

Actually, your spot will be somewhat larger than calculated since spot drills are not ground to a sharp point. If your countersink has tight tolerances, you will have to adjust for it in the program.



Remember, Spot Depth = ½ Spot Diameter

The G81 Drilling Cycle

To use the drilling cycle, follow these programming steps:

1. Load the tool.
2. Position over the first hole.
3. Move to a Z distance above the part while turning on tool length offset.
4. Call the cycle with:

   `G81 X``newx` **`Y`**`newy` **`Z`**`depth` **`F`**`feed`

   Where `newx` and `newy` are the hole position, `depth` is the Z location at the bottom of the hole, and `feed` is the desired feed rate.

The G81 Drilling Cycle (continued)

5. Move to each succeeding hole and drill it.

   Note: All hole cycles are modal, so only the X,Y location of the next hole is needed and the machine will move there and repeat the cycle.

   G80 or G0 may be used to cancel the cycle.

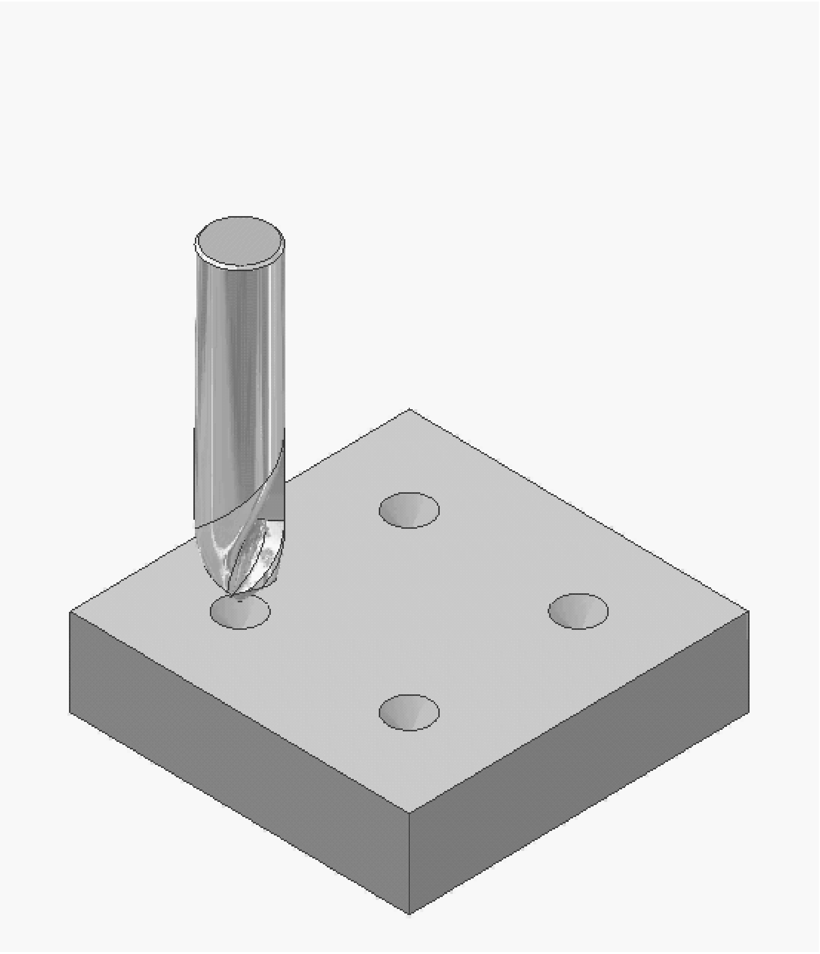
The Spot Drilling Portion of the Program

Program Codes	Action
%	Program Start
O999	
G20 G40 G49 G80 G99	
G91 G28 Z0	
G90	
T12M6	Load tool 12, 5/8” HSS spot drill.
S3667 M3	Set the spindle RPM and direction.
G0 G90 G54 X1.0 Y1.0	Locate over the first hole in the WCS, set fixture offset.
G43 H12 Z0.1 M8	Rapid to clearance with length compensation, coolant on.
G81 Z-0.25 F15.	Drill the hole. The cycle will retract to the initial Z0.1
Y3.0	Move to the next hole and drill it, G81 is modal.
X3.0	Move to the next hole and drill it.
Y1.0	Move to the next hole and drill it.
G80	Cancel the drilling cycle.
.	.
.	The remainder of the program follows.
.	.

Spot Drilling Animation

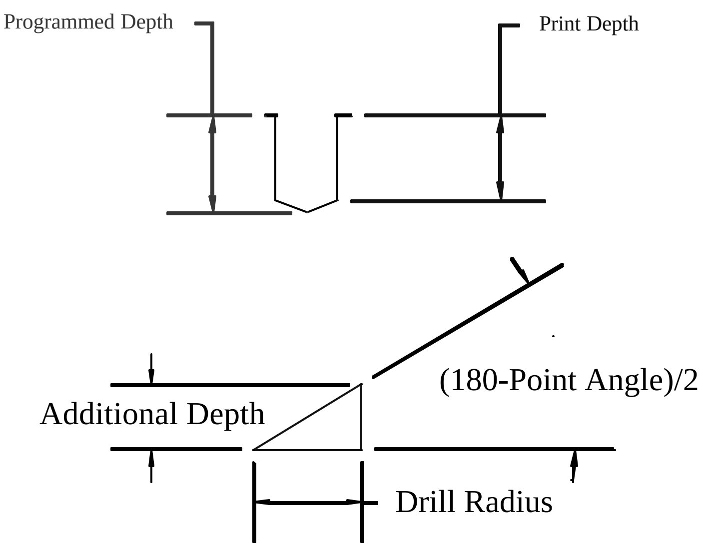
Select this link to
start the animation.

Note: The animation shows the spot drill in its initial position over the first hole and at the initial Z0.1 height. The animation ends at the G80.



Drilling Concerns

- Prints generally give hole depth to the full diameter and do not account for the drill point:
- We must program the depth to the point of the drill. We can calculate the additional depth as follows:



$$\text{Additional Depth} = TAN\left(\frac{180 - \text{PointAngle}}{2}\right) \text{x Drill Radius}$$

For our example with a 0.25” drill and a 118º point angle, the additional depth comes out to 0.075”

The G83 Peck Drilling Cycle

This cycle works much the same as the G81 except for Q, the peck depth:

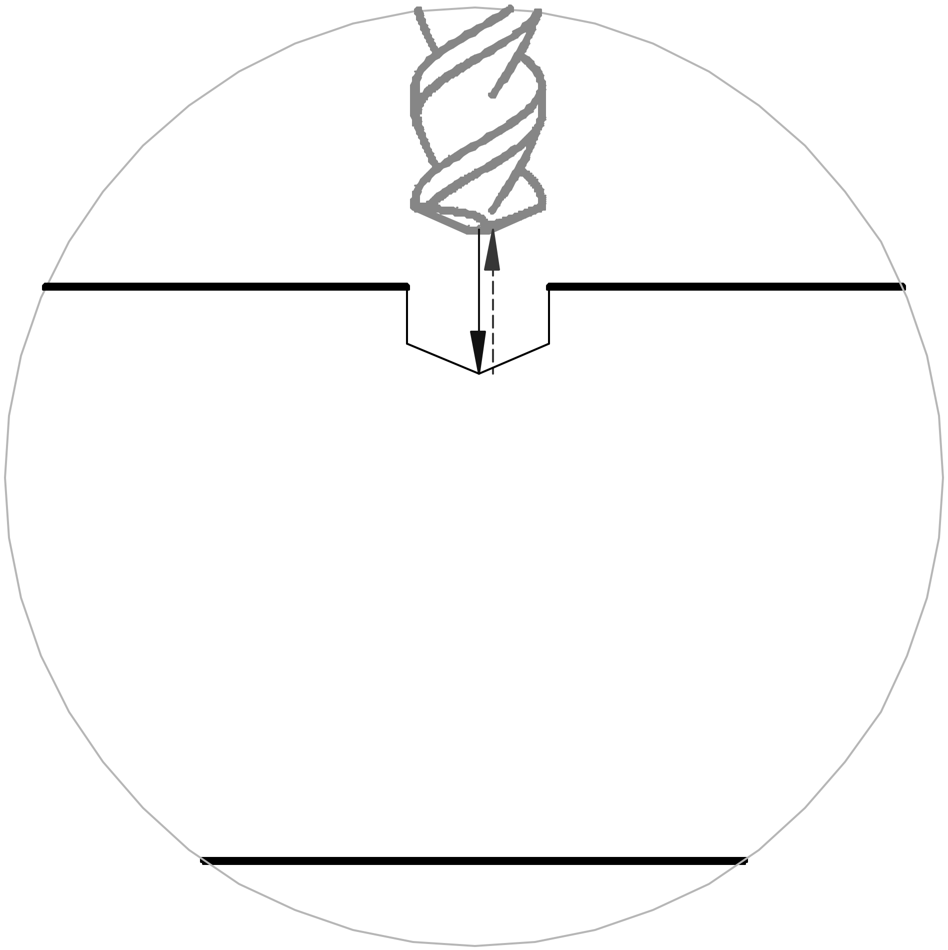
`G83` **`X`**`newx` **`Y`**`newy` **`Z`**`depth` **`Q`**`peckdepth` **`F`**`feed`

Where `newx` and `newy` are the hole position, `depth` is the Z location at the bottom of the hole, `peckdepth` is the amount the drill feeds before backing out of the hole to clear chips, and `feed` is the desired feed rate.

For our example, we will use Q0.25, so it will make four pecks.

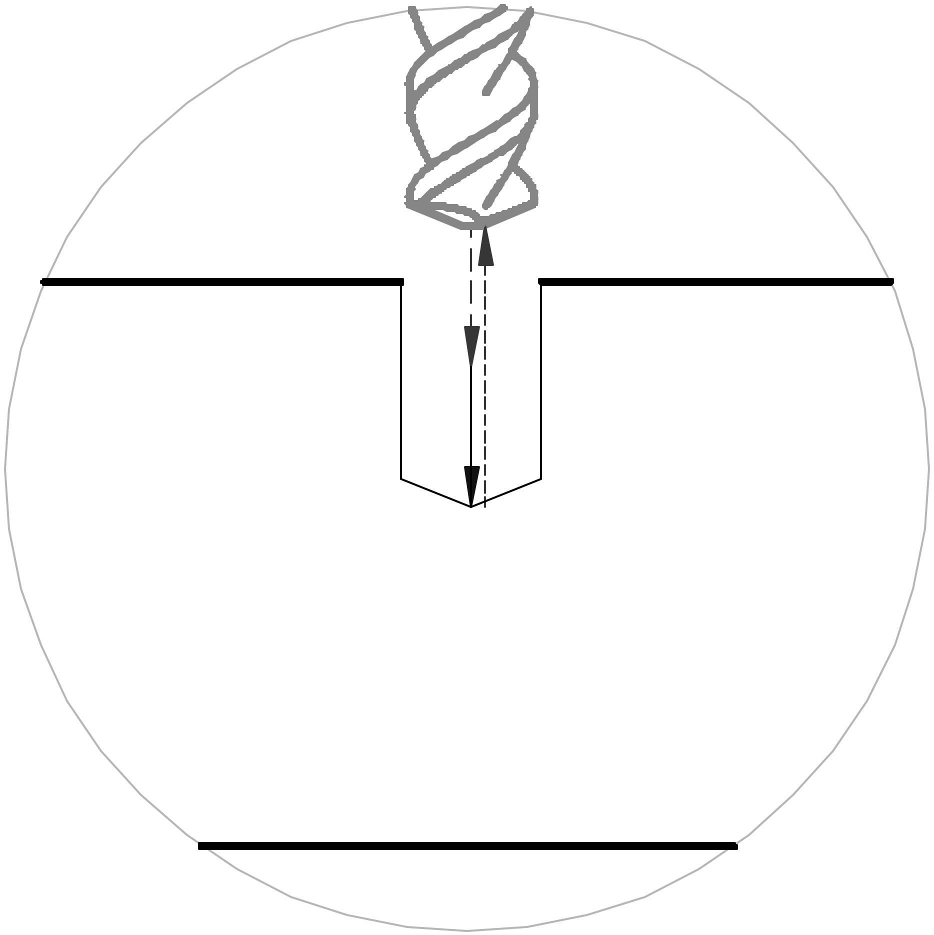
Look at G83 – Peck 1

First Pass 0.25 deep from the initial Z0.1, stops at Z-0.15, then rapids back out to Z0.1.



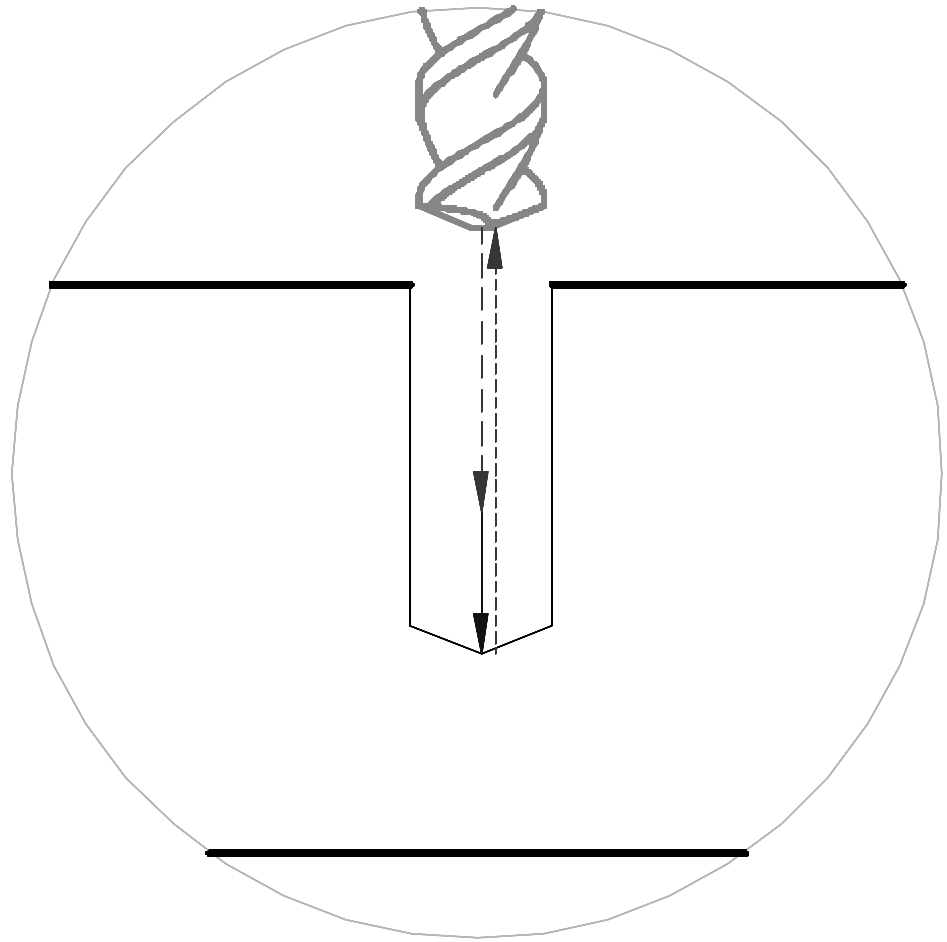
Look at G83 – Peck 2

Second Pass. Rapids just short of previous depth, feeds another 0.25 deep to Z-0.4, then rapids back out to Z0.1.



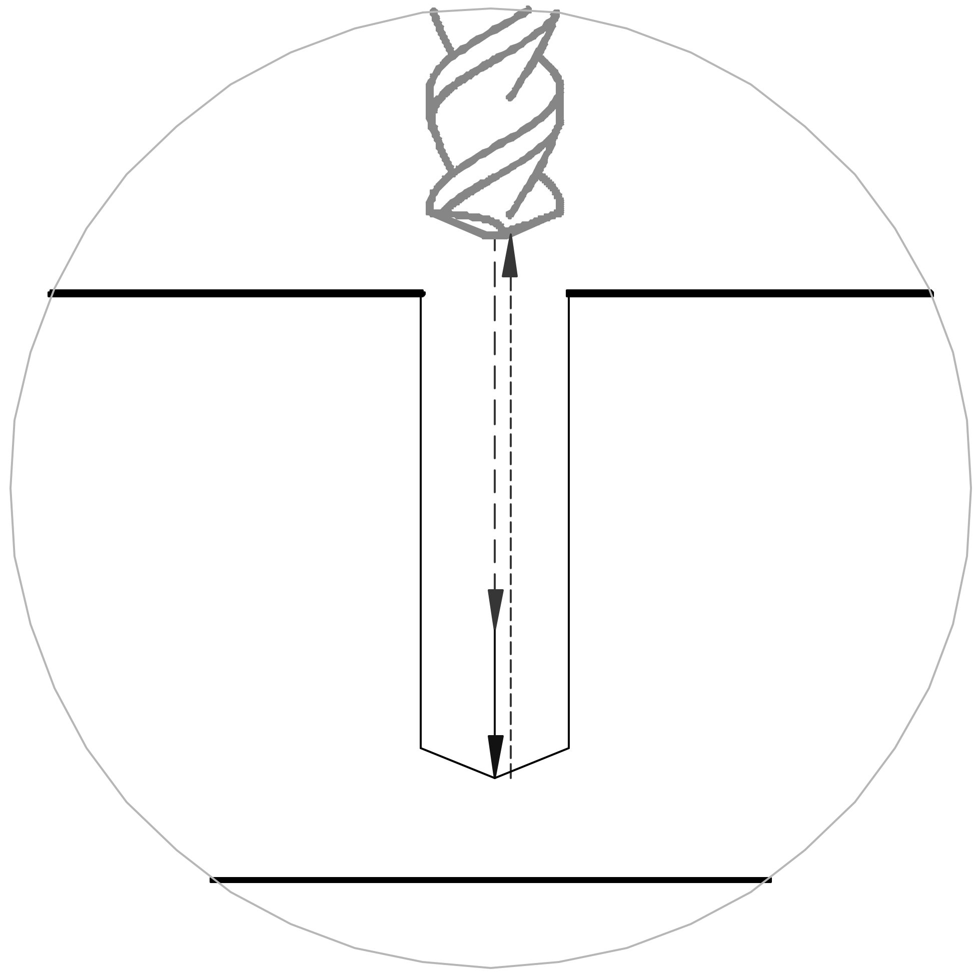
Look at G83 – Peck 3

Third Pass. Rapids just short of previous depth, feeds another 0.25 deep to Z-0.65, then rapids back out to Z0.1.



Look at G83 – Peck 4

Final Pass. Rapids just short of previous depth, feeds to final depth of Z-0.825, then rapids back out to Z0.1.



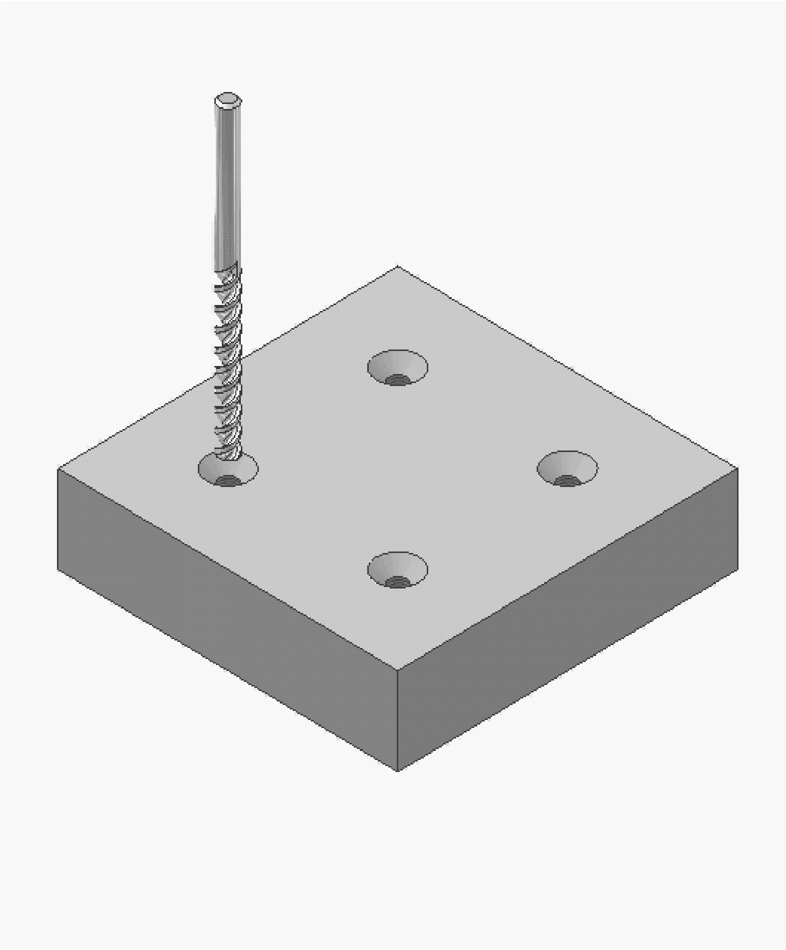
The Peck Drilling Portion of the Program

Program Codes	Action
M9	Tool Change.
M5	
G91 G28 Z0	
G90	
T19M6	Load tool 19, 1/4” HSS drill.
S7000 M3	Set the spindle RPM and direction.
G0 G90 G54 X1.0 Y1.0	Locate over the first hole in the WCS, set fixture offset.
G43 H19 Z0.1 M8	Rapid to clearance with length compensation, coolant on.
G83 Z-0.75 Q0.25 F15.	Drill the hole. The cycle will retract to the initial Z0.1
Y3.0	Move to the next hole and drill it, G83 is modal.
X3.0	Move to the next hole and drill it.
Y1.0	Move to the next hole and drill it.
G80	Cancel the drilling cycle.
.	.
.	The remainder of the program follows.
.	.

Peck Drilling Animation

Select this link to
start the animation.

Note: The animation only shows drilling the first hole to save time. The program will then move the drill to the following holes and it will repeat the drilling cycle. The drill will stop over the last hole at the initial Z0.1 height.



The G84 Right Hand Tapping Cycle

This cycle works much the same as the G81 except the feed must be calculated properly or the threads will strip:

`G84 X``newx` **`Y`**`newy` **`Z`**`depth` **`F`**`feed`

For tapping, the feed rate is calculated with:

$$\text{Feed} = \left(\frac{RPM}{\text{Threads per Inch}} \right)$$

For our example, Feed = 1500/18 = 83.333

Note: be sure to carry three decimal places on Feed when tapping.

Tapping Concerns

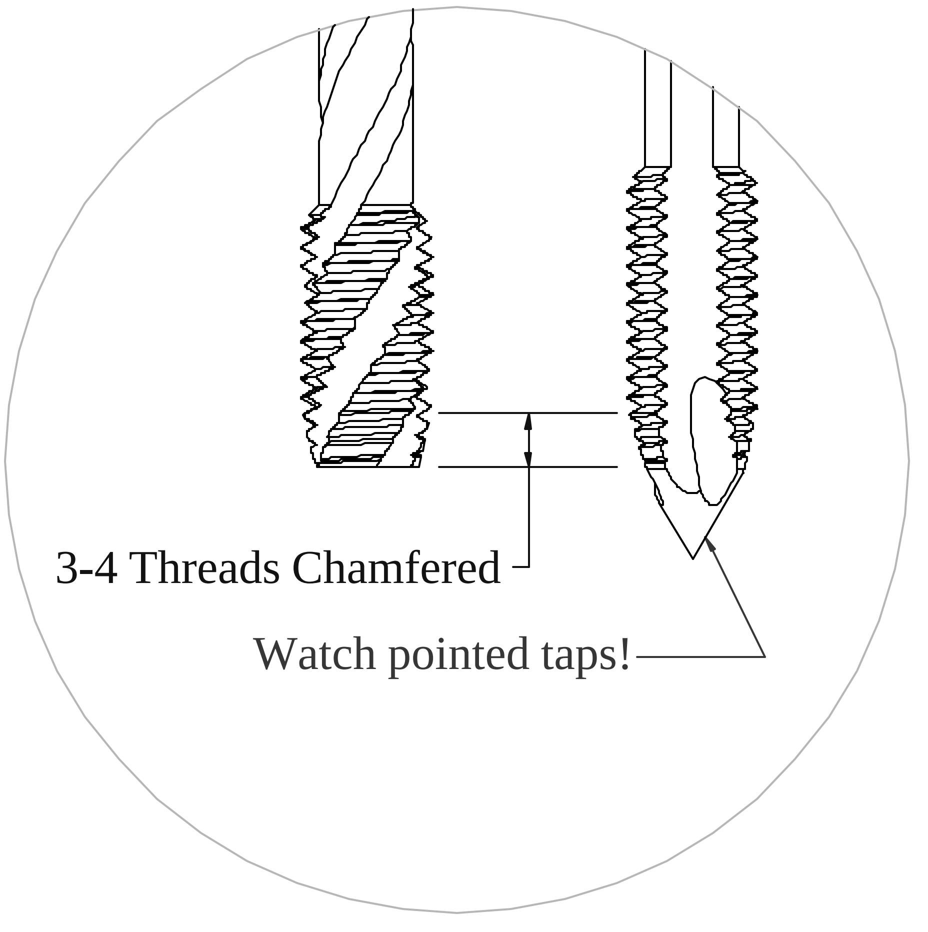
Plug taps are the most common taps used on CNC equipment and they always have 3-4 threads chamfered. If the tap length is set for the end of the tap, this extra distance must be programmed. It is easy to calculate as:

Extra distance = 3/tpi

For our example, this comes to 0.167 which we will round to 0.17".

Be careful of pointed taps! If the tap length is set from the point, then the program must account for this extra depth as well.

Always be sure the tap will not hit the hole bottom!



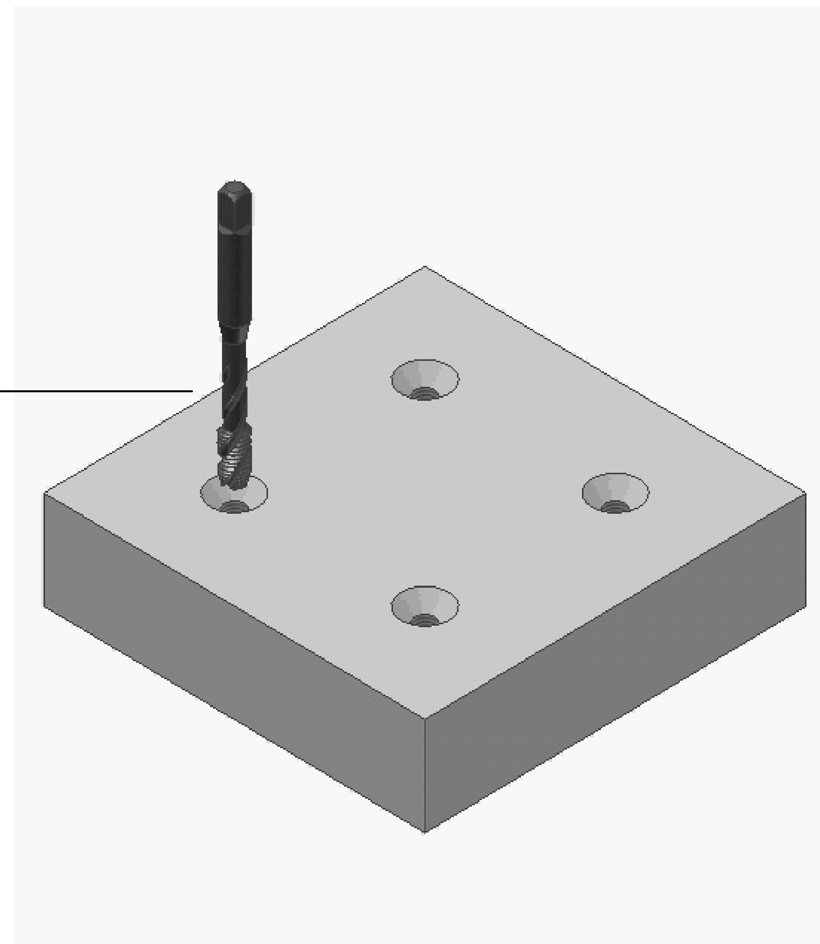
Tapping Portion of the Program

Program Codes	Action
M9	Tool Change.
M5	
G91 G28 Z0	
G90	
T20M6	Load tool 20, 5/16-18 tap.
S1500 M3	Set the spindle RPM and direction.
G0 G90 G54 X1.0 Y1.0	Locate over the first hole in the WCS, set fixture offset.
G43 H19 Z0.1 M8	Rapid to clearance with length compensation, coolant on.
G84 Z-0.67 F83.333	Tap the hole. The cycle will retract to the initial Z0.1
Y3.0	Move to the next hole and tap it, G84 is modal.
X3.0	Move to the next hole and tap it.
Y1.0	Move to the next hole and tap it.
G80	Cancel the cycle.
M9	.
M5	Program End.
G49	.
G91 G28 Z0	
G28 Y0	
G90	
M30	
%	

Tapping Animation

Select this link to start the animation.

Click here to start the animation.

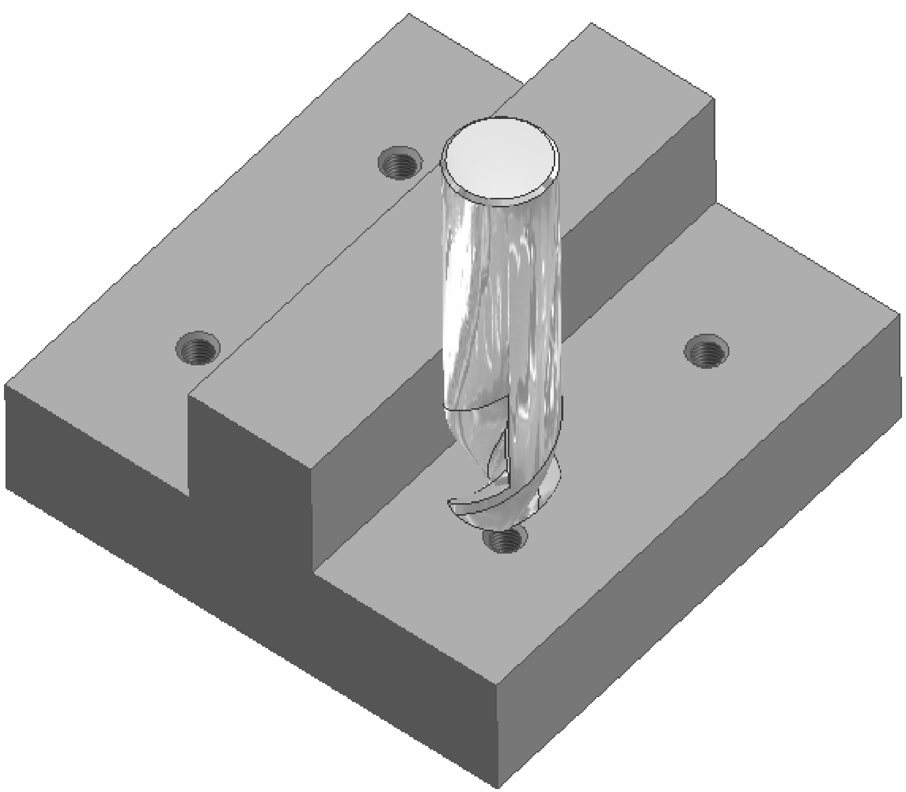


Note: The animation shows the tap in its initial position over the first hole and at the initial Z0.1 height. The animation ends at the G80.

Note that the G84 cycle automatically reverses the spindle and feeds the tap out.

CNC Applications

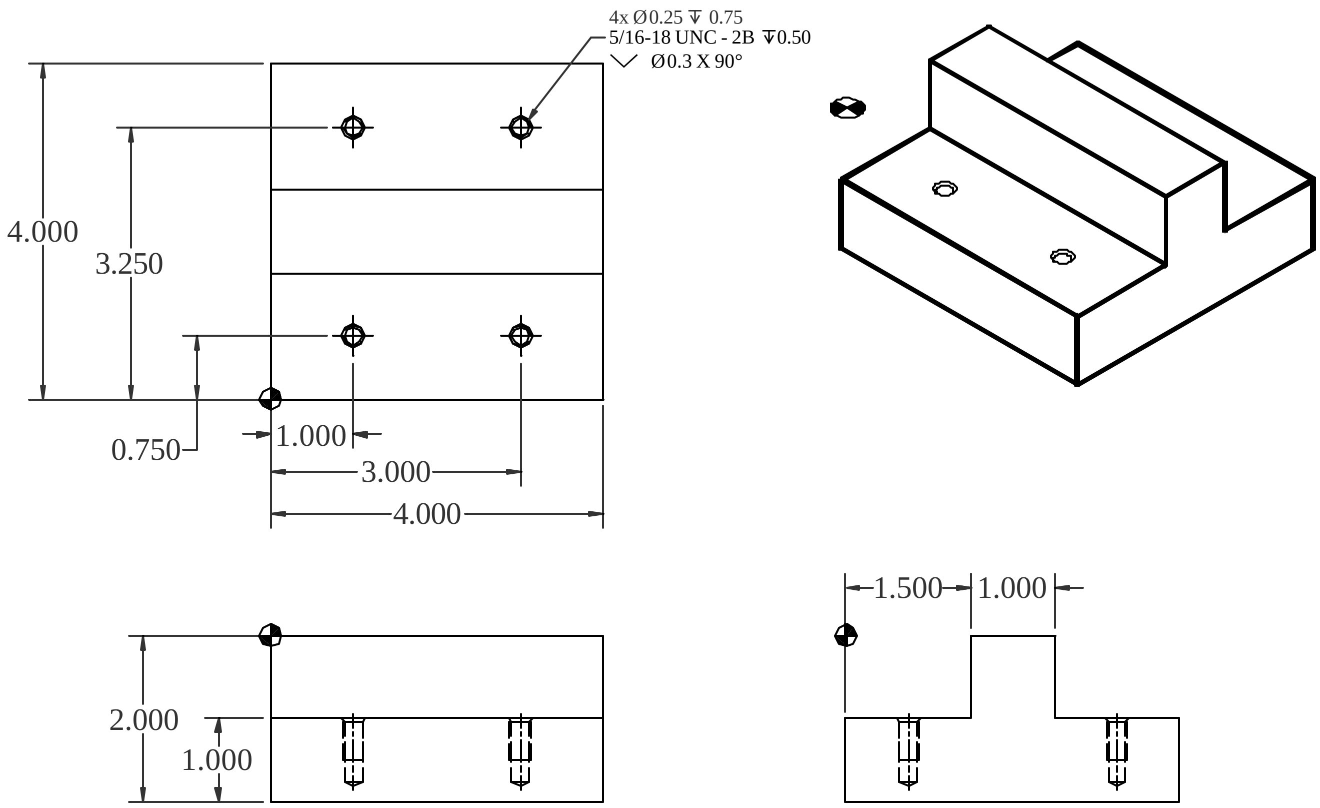
Hole with a Retract Plane



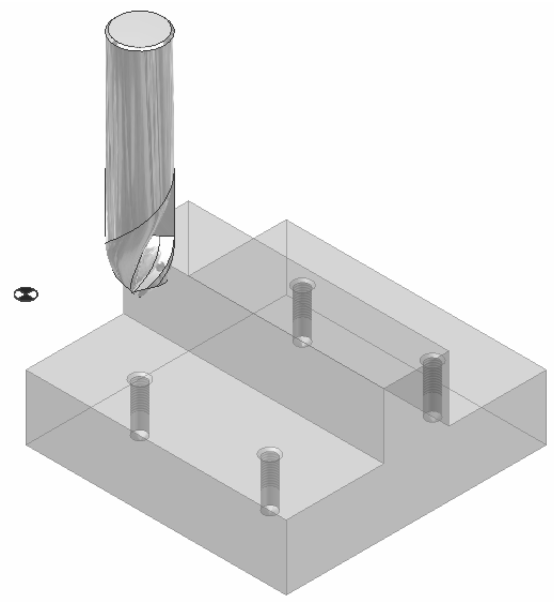
A Common Problem

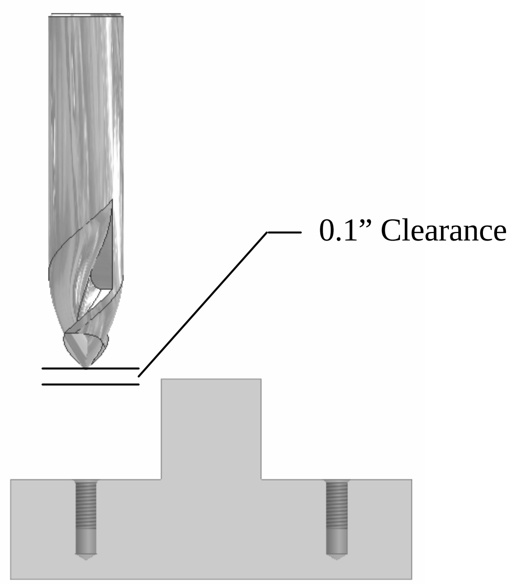
When drilling these holes, we have to stay clear of the boss, but we don't want the drill to start feeding above the part.

There is an easy solution to this problem – use an R plane.



The Initial or I Plane

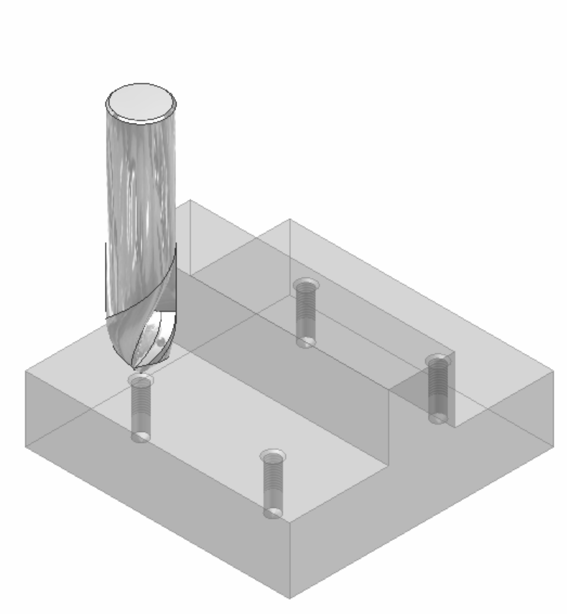


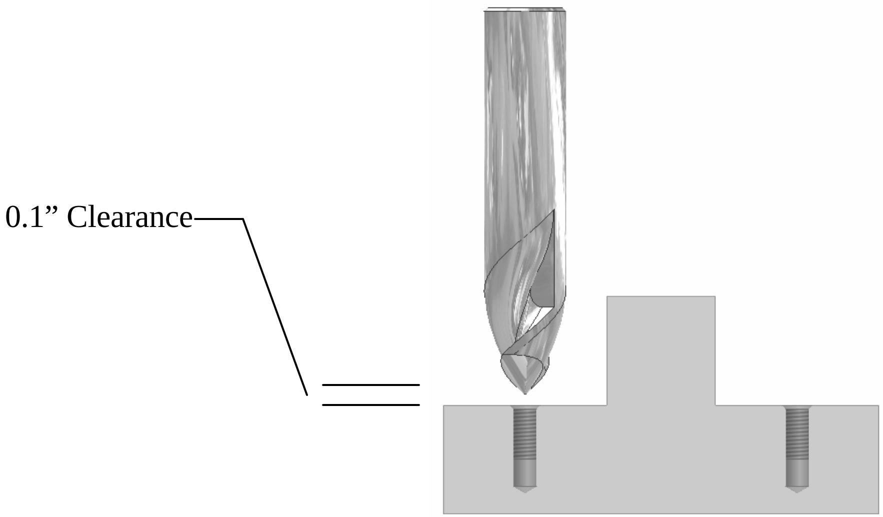


The I Plane is the Z location we first bring the tool down to **before** initiating a hole cycle. We generally use 0.1” above the part. We normally use a G0 to position the cutter in this position. For our examples, it is these blocks:

```
G0 G90 G57 X1.0 Y0.75
G43 H12 Z0.1 M8
```

The Rapid or R Plane





The R Plane is the Z location we rapid the tool down to **in the hole cycle** before feeding starts. We generally use 0.1” above the surface the hole is at, Z-0.9 for this example.

Programming the R Plane

To add an R plane to any hole cycle, we just add R with the Z value to the cycle. For this example:

```
G81 Z-1.15 R-0.9 F15.
```

This works for all cycles as follows:

```
G83 Z-1.75 Q0.25 R-0.9 F15.

G84 Z-1.67 R-0.9 F83.333
```

The only problem is that holes on the same side of an obstruction cause wasted time because the cutter rapids up and down from the R plane. To solve this problem, we use G98 and G99.

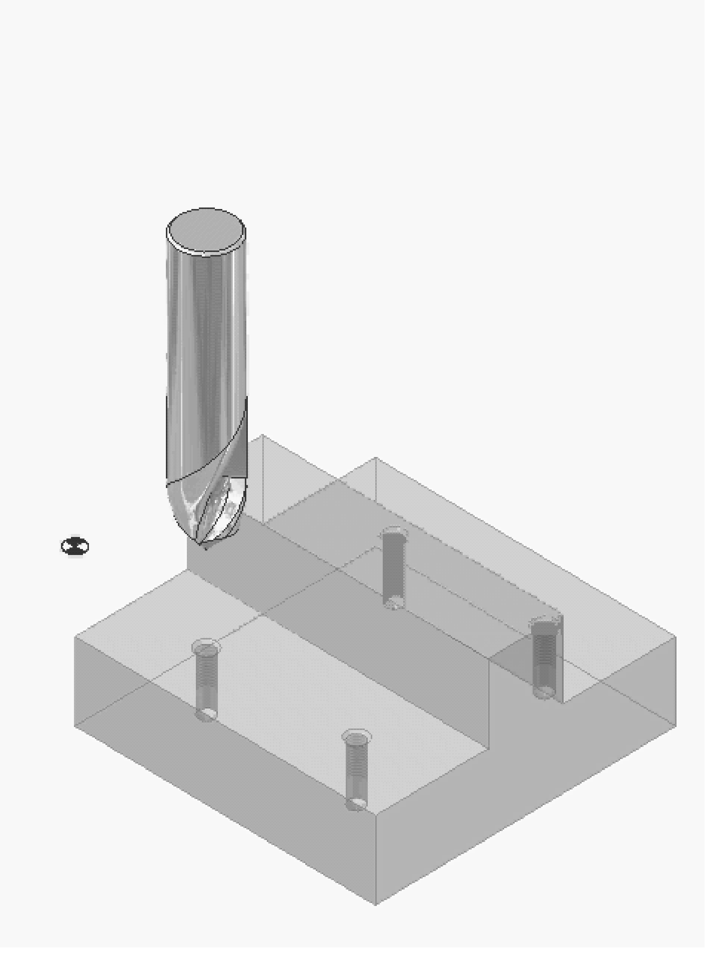
G Code	Application
G98	After drilling this hole, return to the I plane. This is usually the default.
G99	After drilling this hole, return to the R plane.

The Spot Drilling Portion of the Program with R Plane

Program Codes	Action
%	Program Start
O999	
G20 G40 G49 G80 G99	
G91 G28 Z0	
G90	
T12M6	Load tool 12, 5/8" HSS spot drill
S3667 M3	Set the spindle RPM and direction
G0 G90 G57 X1.0 Y0.75	Locate over the first hole in the WCS, set fixture offset.
G43 H12 Z0.1 M8	Rapid to the I plane with length compensation, coolant on
G81 Z-1.15 R-0.9 F15. G99	Drill the hole. The cycle will retract to the R plane Z=-0.9
X3.0 G98	Drill the next hole, retract to I plane Z=0.1 to clear part.
Y3.25 G99	Retract to the R plane Z=-0.9
X1.0 G98	Retract to the I plane Z=-0.1
G80	Cancel the drilling cycle.
.	.
.	The remainder of the program follows.
.	.

Spot Drilling Animation with R Plane

Select this link to start the animation.



The animation starts with the spot drill at Z0.1 and ends with the G80. Notice how the drill only retracts to Z-0.9 between holes on the same side of the obstruction.

The Peck Drilling
Portion of the Program with R Plane

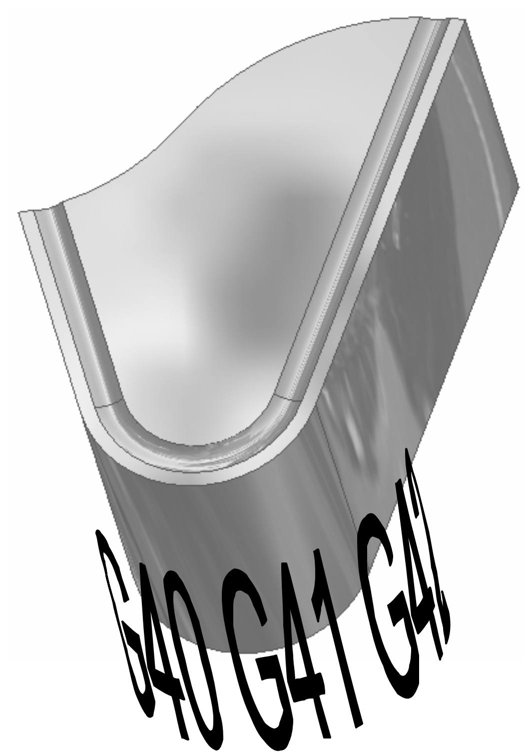
Program Codes	Action
M9	Tool Change.
M5	
G91 G28 Z0	
G90	
T19M6	Load tool 19, 1/4” HSS drill
S7000 M3	Set the spindle RPM and direction
G0 G90 G57 X1.0 Y0.75	Locate over the first hole, set fixture offset.
G43 H19 Z0.1 M8	Rapid to the I plane with length compensation
G83 Z-1.75 Q0.25 R-0.9 F15. G99	The cycle will retract to the R plane, Z=-0.9
X3.0 G98	The cycle will retract to the I plane, Z0.1
Y3.25 G99	Retracts to R plane
X1.0 G98	Retracts to I plane
G80	Cancel the drilling cycle.
.	.
.	The remainder of the program follows.
.	.

Tapping Portion of the Program

Program Codes	Action
M9	Tool Change.
M5	
G91 G28 Z0	
G90	
T20M6	Load tool 20, 5/16-18 tap.
S1500 M3	Set the spindle RPM and direction.
G0 G90 G57 X1.0 Y0.75	Locate over the first hole in the WCS.
G43 H19 Z0.1 M8	Rapid to clearance with length compensation.
G84 Z-1.67 R-0.9 F83.333 G99	Tap the hole. The cycle will retract to the R plane, Z-0.9
X3.0 G98	Move to the next hole and tap it, retracts to I plane, Z0.1
Y3.25 G99	Retracts to R plane, Z-0.9
X1.0 G98	Retracts to I plane, Z0.1
G80	Cancel the cycle.
M9	.
M5	Program End.
G49	.
G91 G28 Z0	
G28 Y0	
G90	
M30	
%	

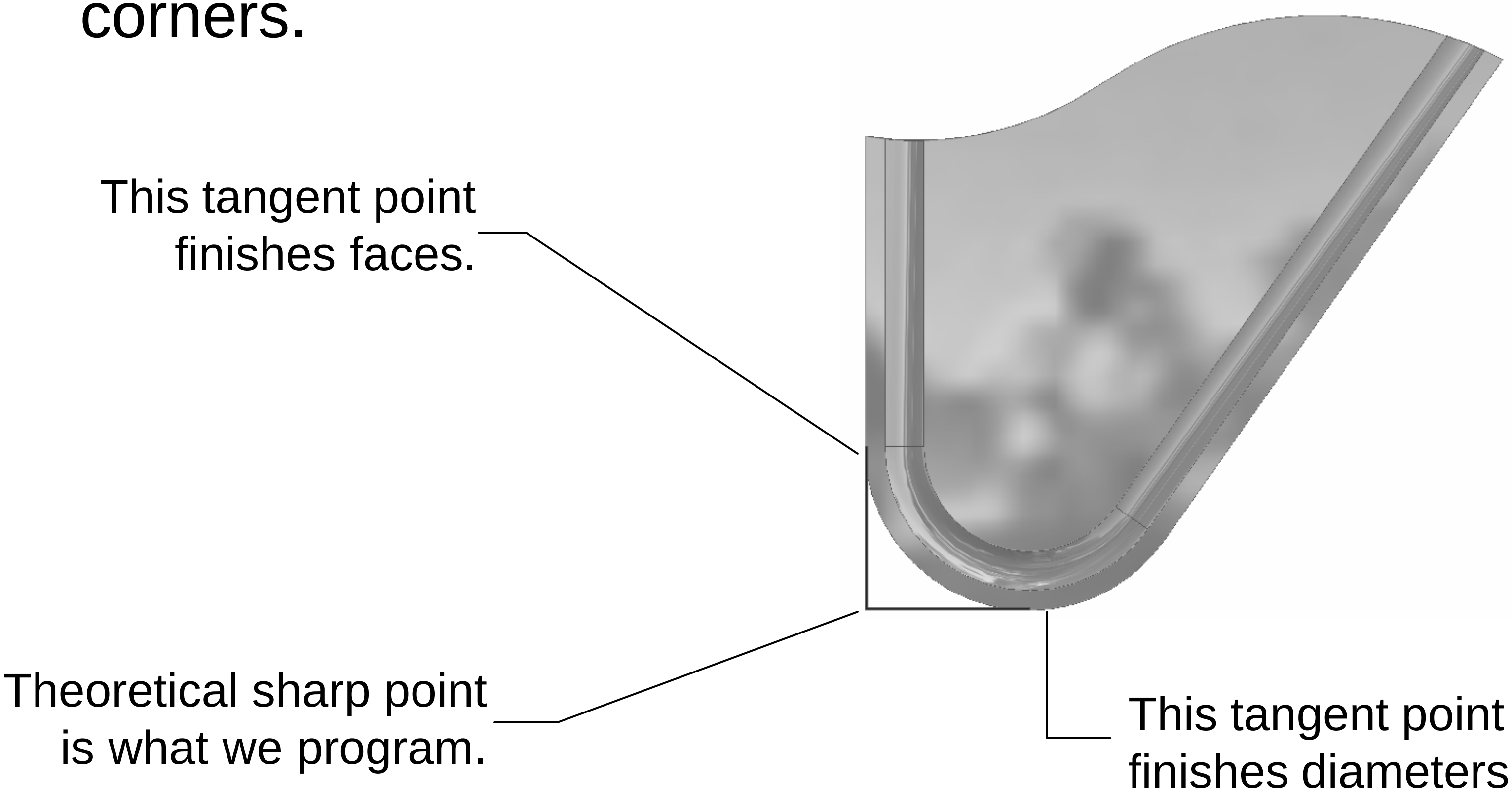
CNC Applications

Tool Nose Radius Compensation on Turning Centers



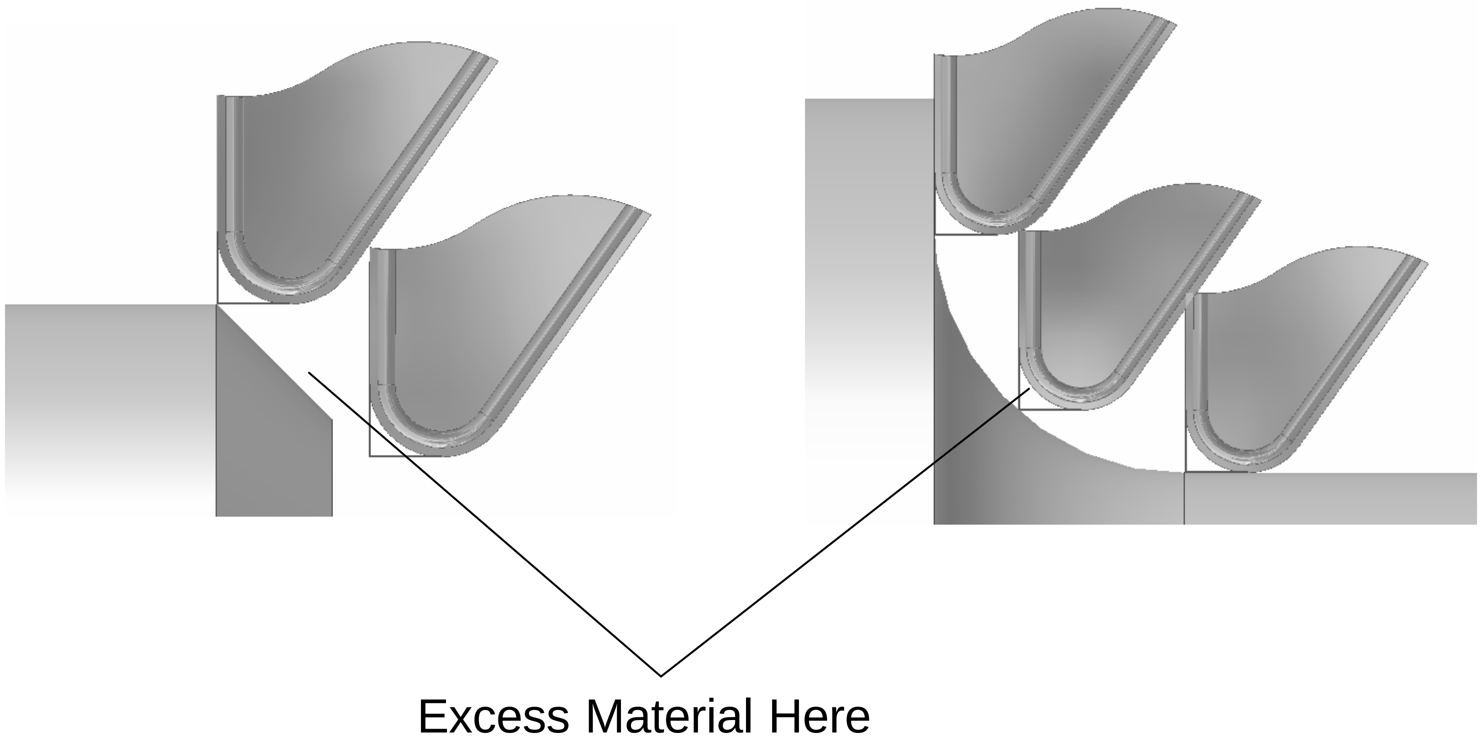
Facing and Straight Turning

- When facing or straight turning, the tool nose radius has no effect on the part other than leaving a radius on inside corners.



The Problem

When turning tapers or radii, the tool nose radius leaves excess material as shown here:



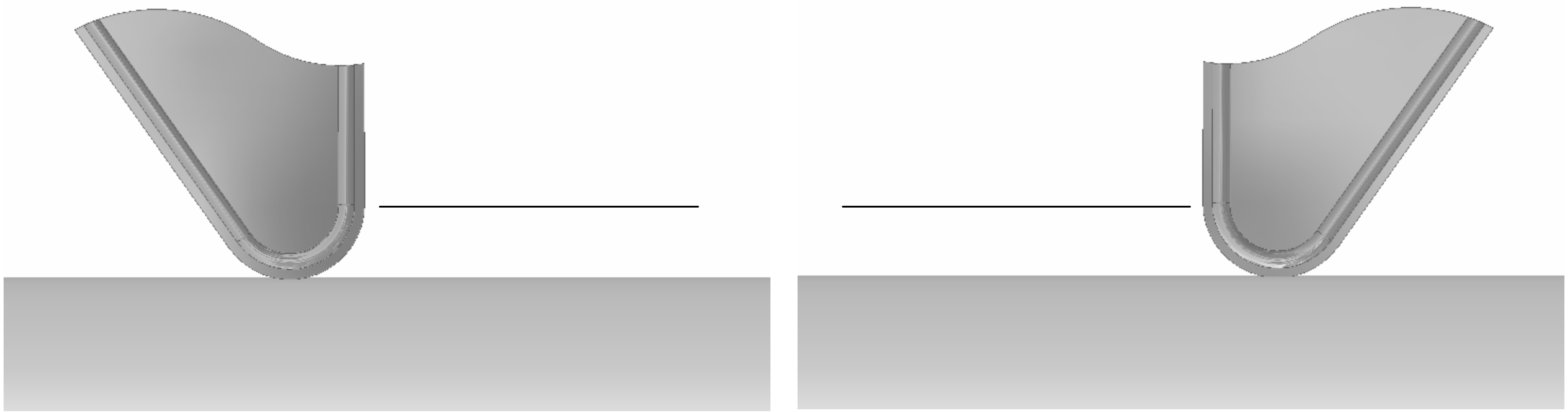
The Solution

1. Manually program the exact tangent points. This is time consuming since it requires trig calculations or accurate CAD drawings to locate the tangent points.
2. Use tool nose radius compensation. The tool nose radius is entered into the machine controller, and the program turns on compensation for finish cuts only, and then turns it off. The machine calculates the tangent points so we can continue programming as if the cutter has a sharp point.

Tool Nose Radius G Codes

G Code	Application
G40	Cancel tool nose radius compensation.
G41	Compensate for tool nose radius to the LEFT of the programmed path.
G42	Compensate for tool nose radius to the RIGHT of the programmed path.

G41 & G42

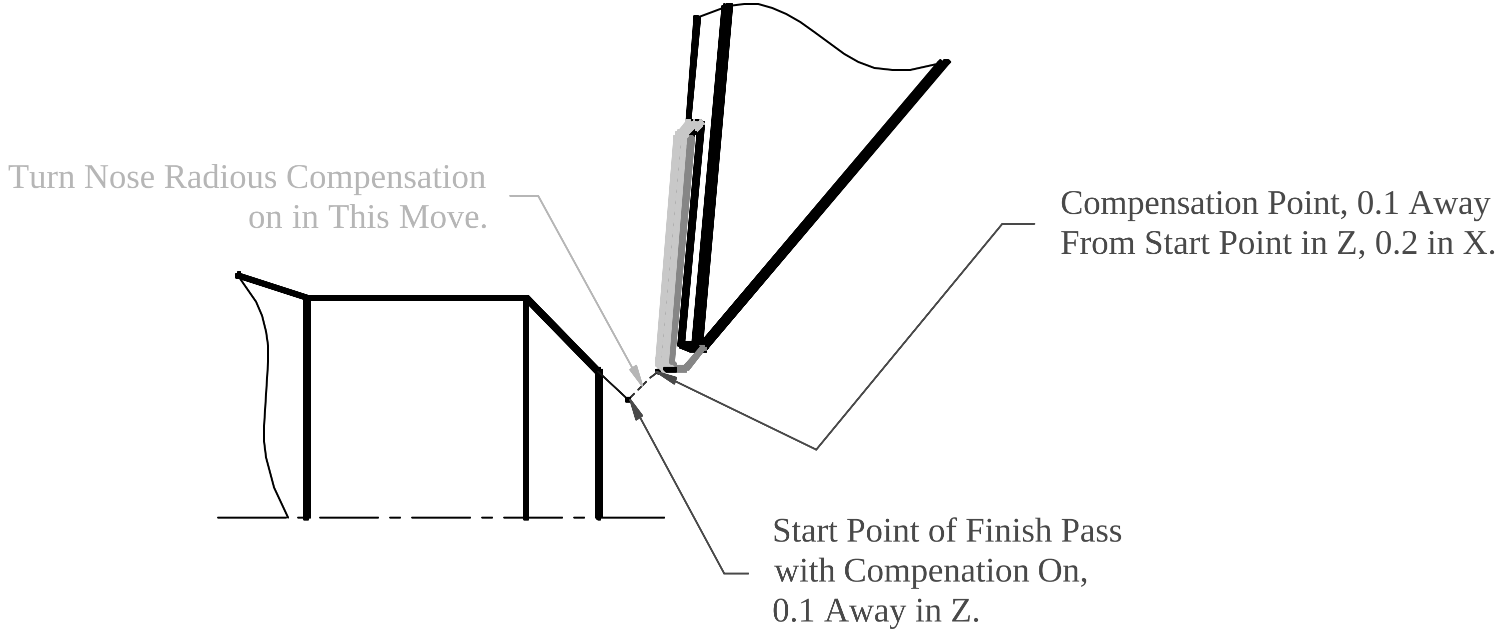


G41 – the cutter is to the left of the work when looking in the direction of the cut.

G42 – the cutter is to the right of the work when looking in the direction of the cut.

Turning Nose Radius Compensation On

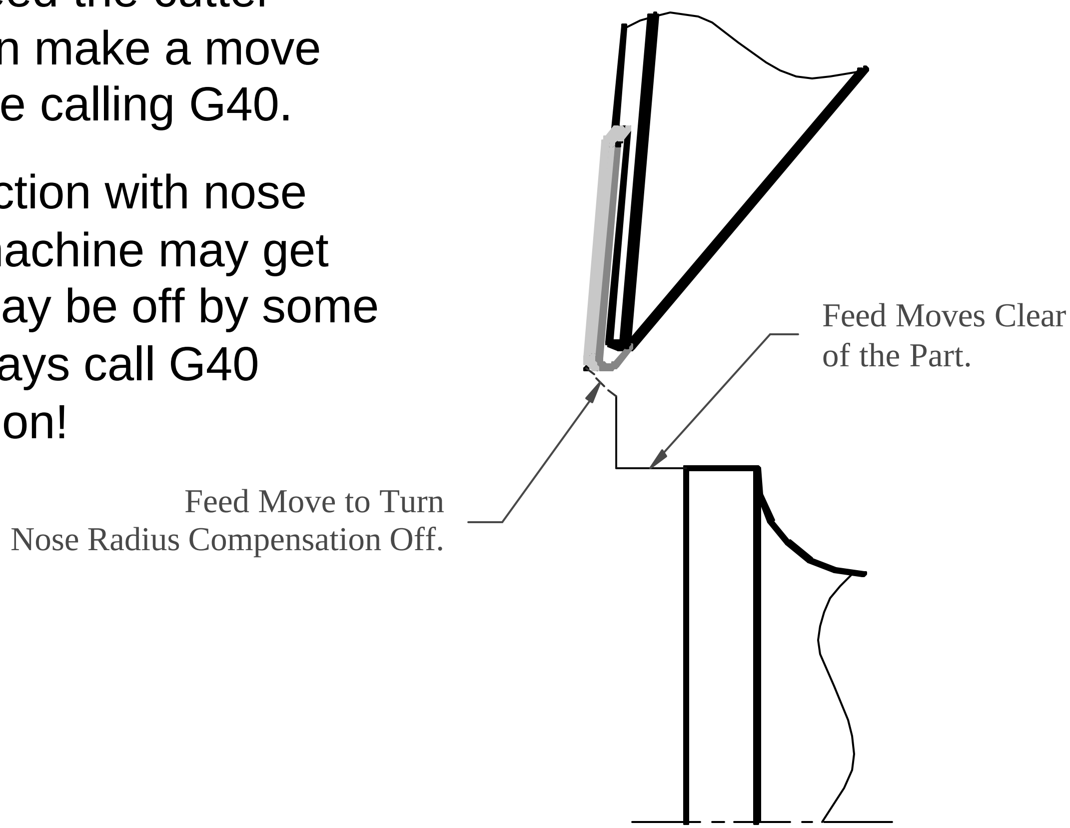
To turn compensation on, the machine must move at least the distance of the nose radius in X and Z. For easy calculations, back away from the start point 0.1 in Z and 0.2 in X. Remember X is diameter based, so 0.2 in X is actually 0.1 radially.



Turning Nose Radius Compensation Off

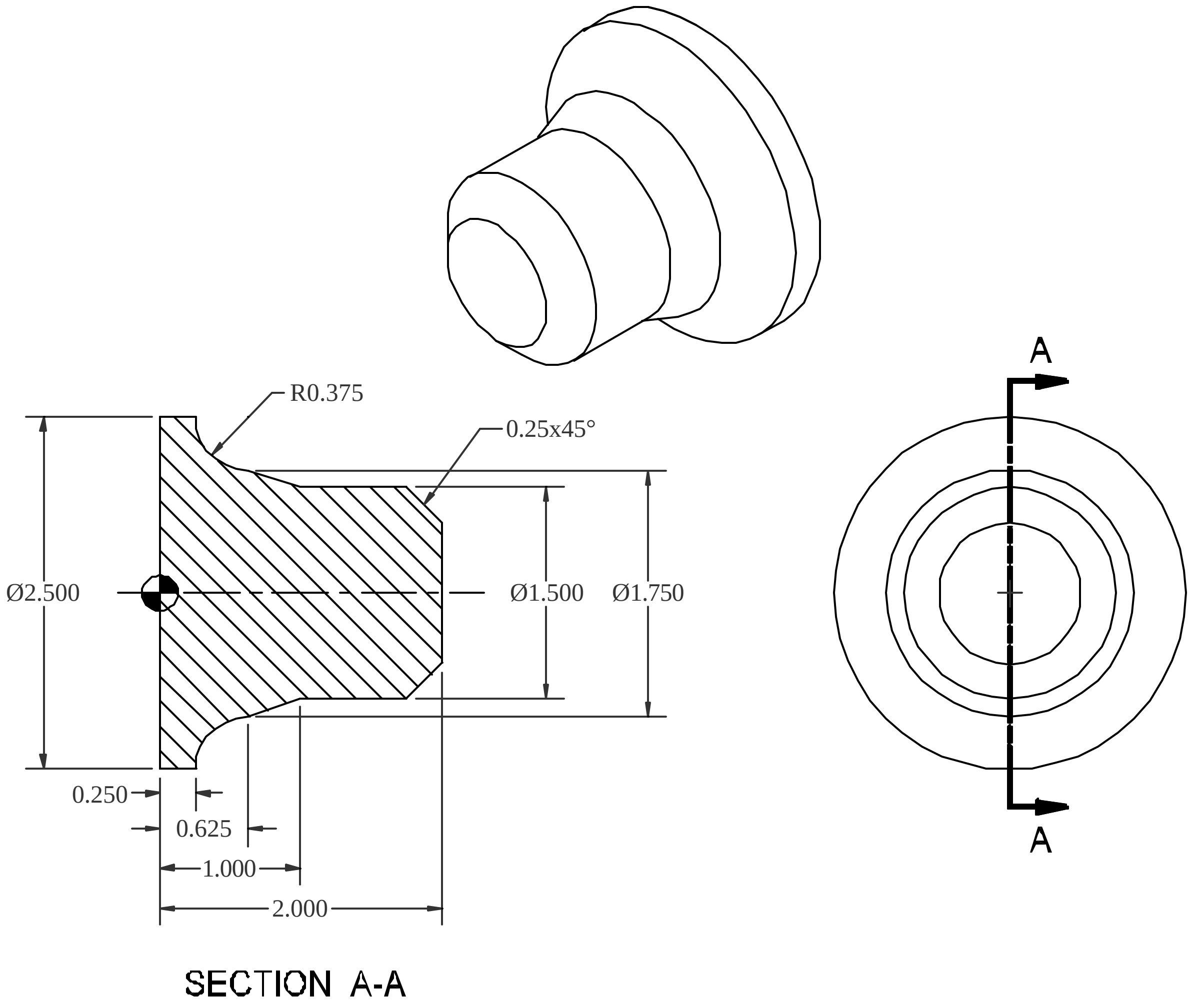
To turn compensation off, we feed the cutter completely off the work and then make a move larger than the nose radius while calling G40.

Note: Do not reverse the Z direction with nose radius compensation on! The machine may get confused, and then later cuts may be off by some multiple of the nose radius. Always call G40 BEFORE reversing the Z direction!



A G42 Example

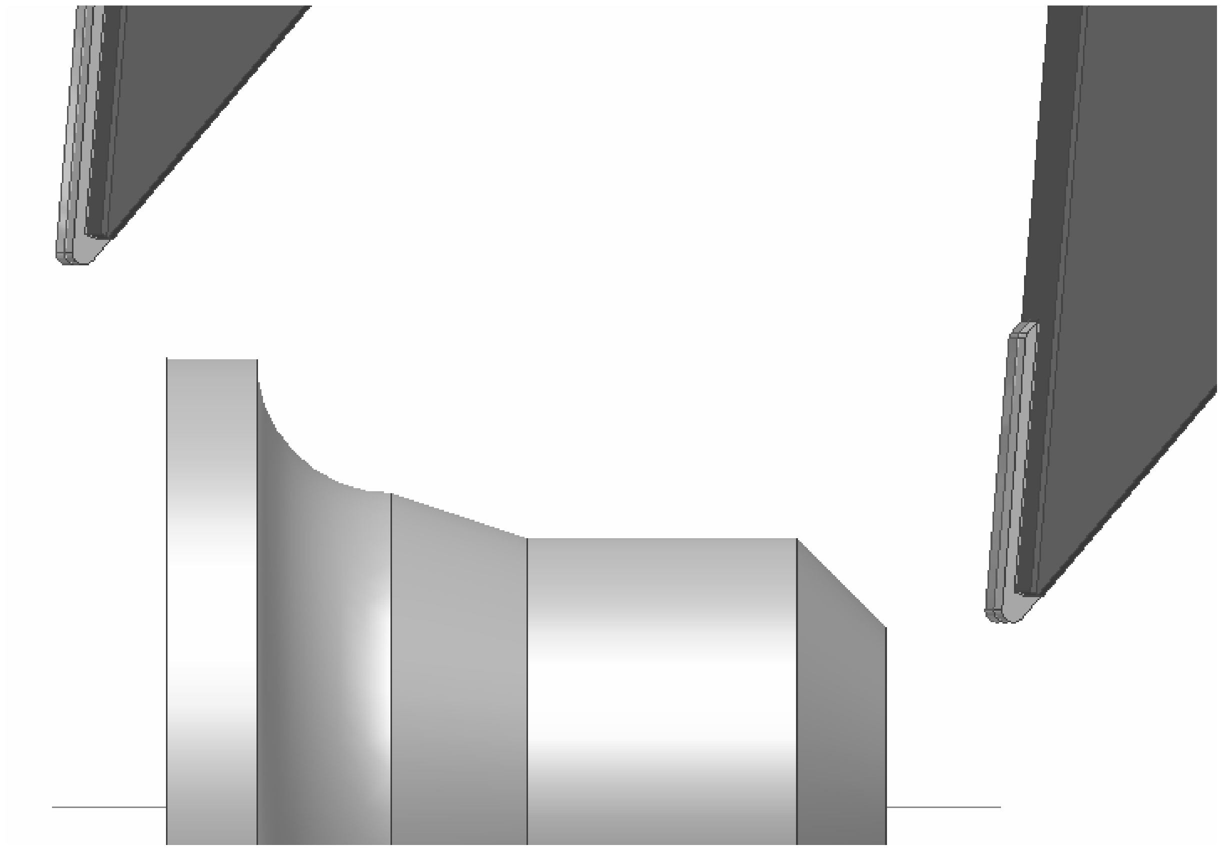
We will program ONLY the finish pass on this part using G42 right tool nose radius compensation. We are given 800fpm cutting speed and 0.006ipr feed.



The Finish Pass

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0303	Load the V insert tool.
G54	
G50 S4000	Cap the RPM.
G96 S800 M3	Set the cutting speed to 800fpm, forward direction.
G0 Z2.2	Rapid to the G42 start point in Z.
X1.0 M8	Rapid to the G42 start point in X, coolant on.
G42 G1 X0.8 Z2.1 F0.006	Move to turn nose radius compensation on, beginning of chamfer.
X1.5 Z1.75	Machine the chamfer.
Z1.0	Machine the straight 1.0” diameter.
X1.75 Z0.625	Machine the taper.
G2 X2.5 Z0.25 I0.375	Machine the radius.
Z-0.15	Feed clear in Z leaving room for the 0.125” parting tool.
X2.875	Feed clear in X.
G40 X3.075 Z-0.25	Move to turn off nose radius compensation.
M9	Program End
M5	
G28 U0	
G28 W0	
M30	
%	

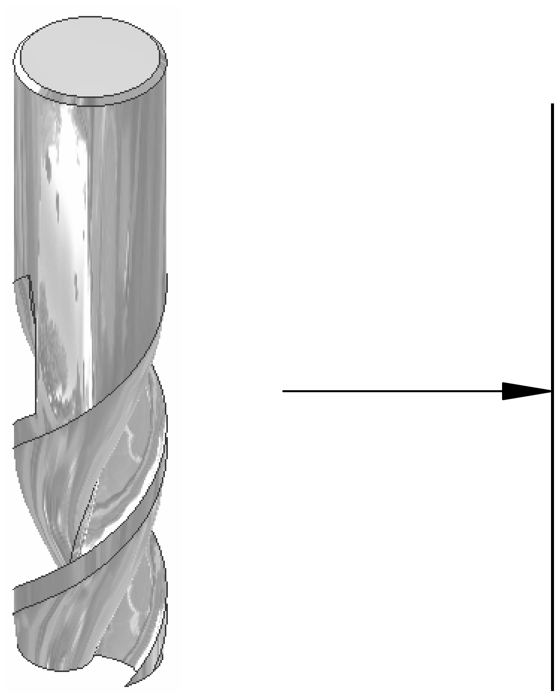
The Final Pass



Select this link to start the animation.

CNC Applications

Tool Radius Compensation for Machining Centers



Why Cutter Diameter Compensation?

- When machining finished surfaces with the side of a milling cutter (generally called profiling), the accuracy of the finished surface depends on the cutter accuracy and how closely the cutter diameter matches the programmed size.
- Cutters wear causing size changes in profiled surfaces.
- Reground endmills are always smaller than nominal size.

Note: this feature is also frequently called Cutter Radius Compensation. We use Diameter Compensation to avoid confusion with turning center operation.

How Does Cutter Diameter Compensation work?

- The programmer programs for a nominal size cutter (0.750” for our T1, for example) for rough cuts.
- The programmer calls cutter diameter compensation for THE FINISH PASS ONLY, and programs as if the cutter has zero diameter.
- The set-up person enters the nominal size in the machine controller.
- Parts are measured as they are manufactured, and the operator enters small deviations to keep the parts on size (generally called wear offsets). The machine automatically adjusts for the wear offsets.

Cutter Diameter Compensation Codes

Code	Application
G40	Cancel cutter diameter compensation.
G41	Compensate for the cutter to the LEFT of the programmed path.
G42	Compensate for the cutter to the RIGHT of the programmed path.
Dtt	tt is the tool number. D tells the controller where to find the cutter's diameter.

Determining G41 or G42

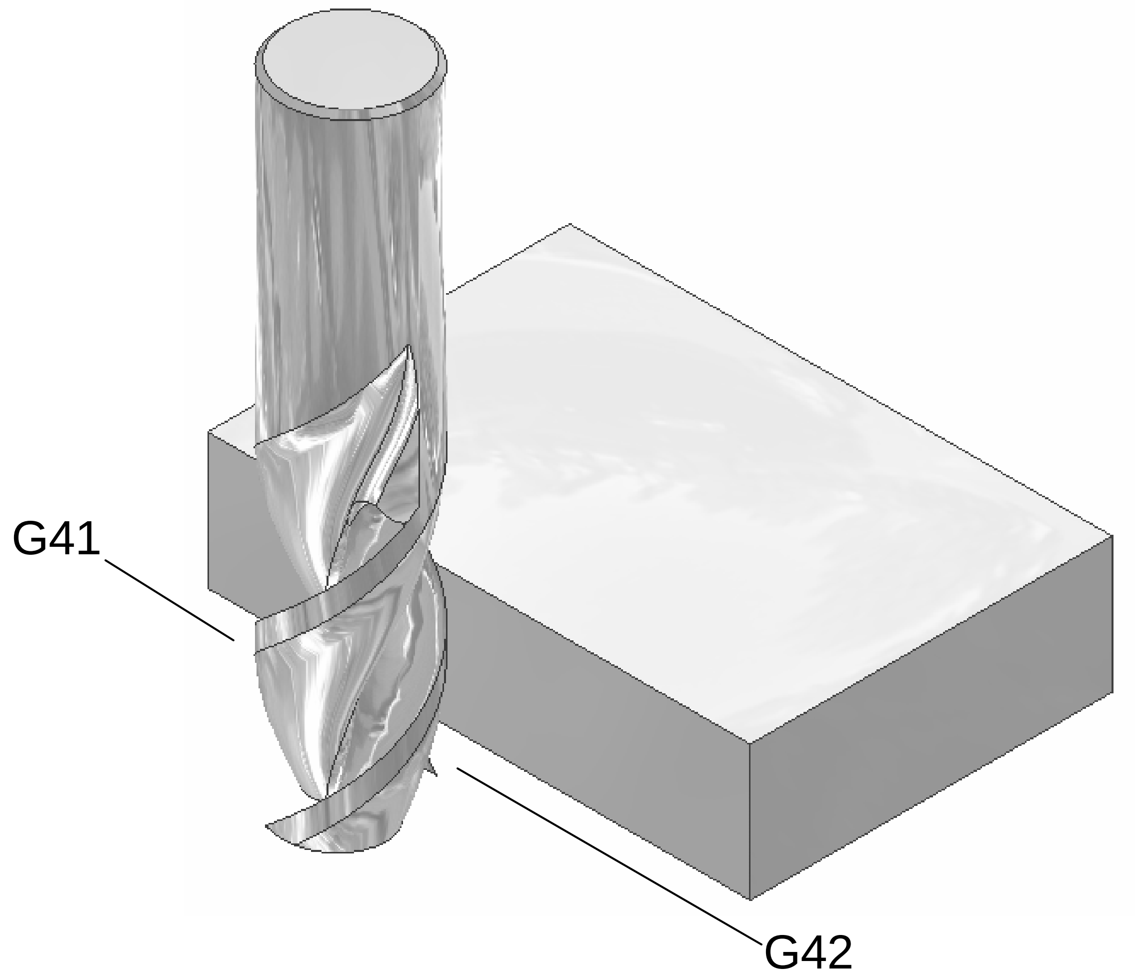
G41: the cutter is to the left of the part when looking in the direction of the cut.

G42: the cutter is to the right of the part when looking in the direction of the cut.

Climb milling features: use G41.

Conventional milling features: use G42.

Since we normally climb mill, we will generally use G41 on a machining center.



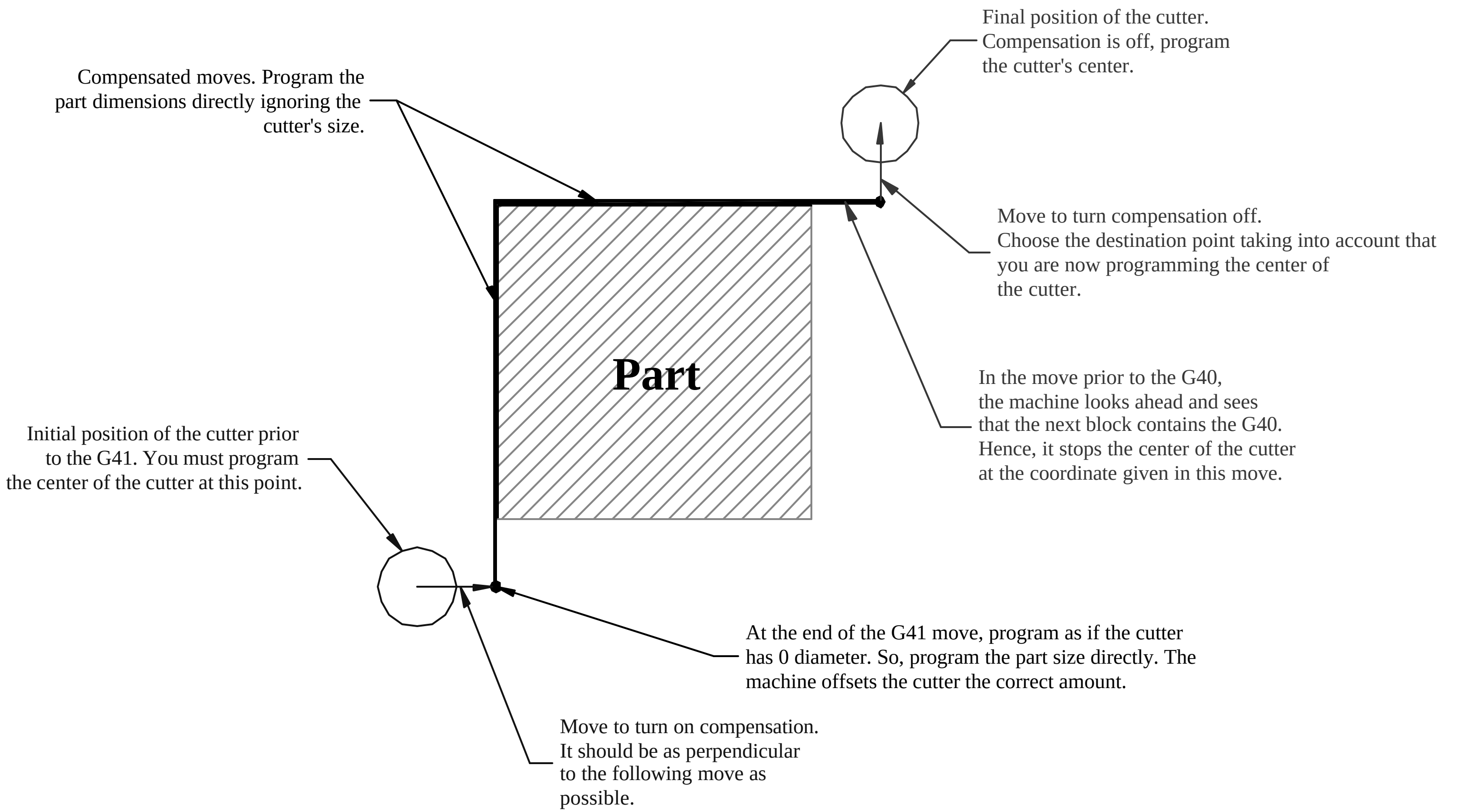
Turning Cutter Diameter Compensation On

- Prior to compensation, you must take into account the cutter’s radius and locate the cutter offset by its radius.
- In the move turning compensation on, try to make the move as perpendicular to the following move as possible, and try to turn compensation on with the cutter off the part. Note: the D code should be in the same block as the G41 or G42.
- The move to turn compensation on should be equal or greater than the cutter’s radius.
- Once compensation is on, ignore the cutter’s size and program as if the cutter has 0 diameter. This usually simplifies programming, especially arc programming.

Turning Cutter Diameter Compensation Off

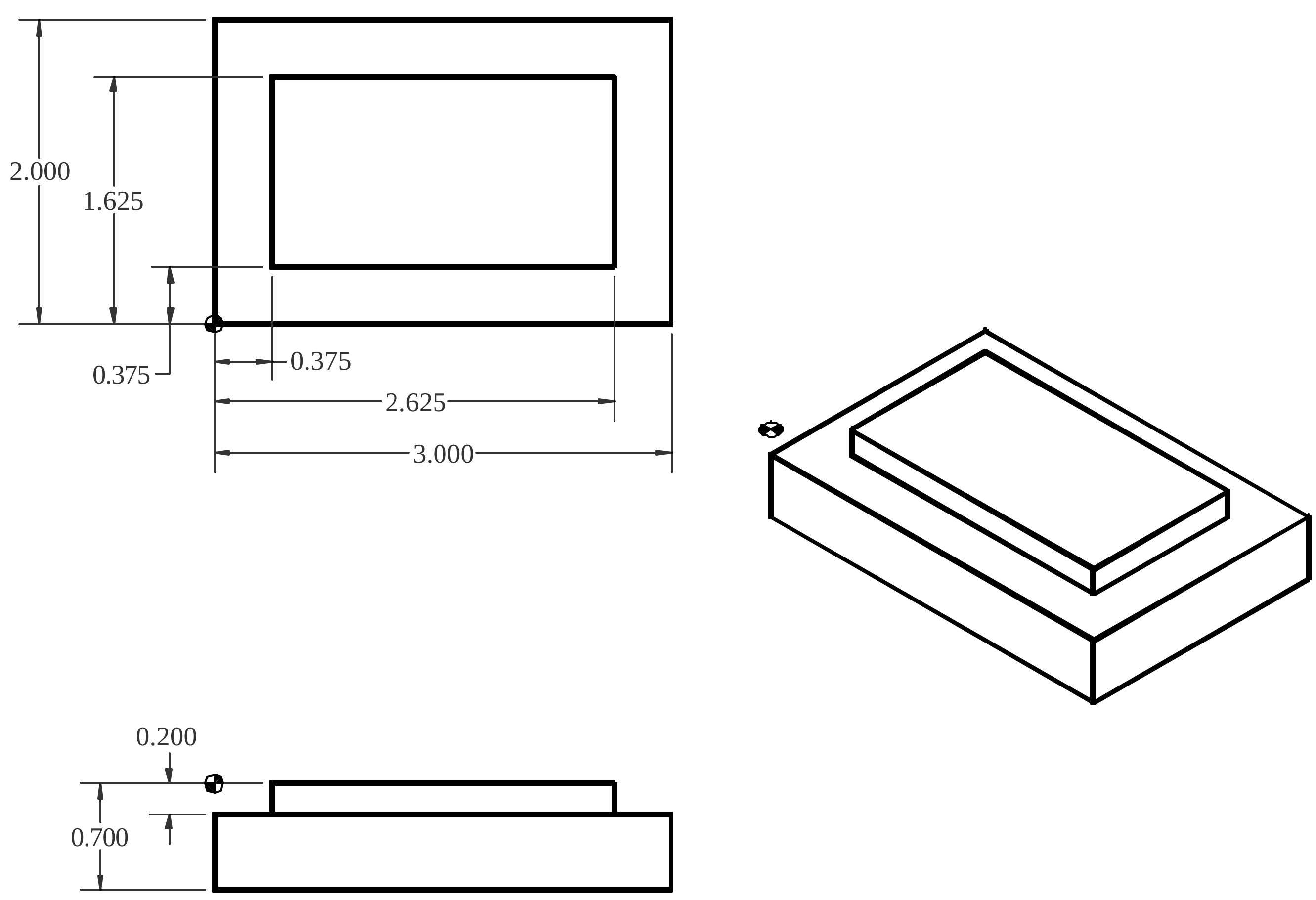
- When performing cutter compensation, the machine looks ahead in your program several blocks so it can calculate tangent points.
- In the move prior to turning compensation off, the machine moves the cutter to the center point in the direction of this move, and then to the cutter’s center point in the direction of the G40 move.
- If possible, turn compensation off with the cutter off the part. If that is not possible, try to select the end point in a clear area of the part such as the center of a pocket.

A Graphical Look at Cutter Diameter Compensation



A Compensation Example

We’ll program this part with a ¾” endmill, T1. We’ll assume the length and thickness are already machined. So, we will take one roughing pass leaving 0.010” on the profile, and then we will finish the profile with tool diameter compensation.



Follow Planning and Programming Steps (1-5)

1. Examine drawing.
2. How will we hold the raw material – in a vise up on parallels.
3. Decide what cutters to use – given a ¾” HSS endmill. We previously calculated 3056RPM and 24IPM.
4. Write down the exact sequence of operations:
   A. Rapid position the cutter clear in –Y.
   B. Rough machine the profile leaving 0.010” material.
   C. Position the cutter clear of the part.
   D. Turn on compensation, finish machine the profile.
   E. Turn off compensation.
   F. Program end.
5. Convert the sequence of operations to a program:

Program Start

Rough Machining

Finish Machining with Compensation

Program End

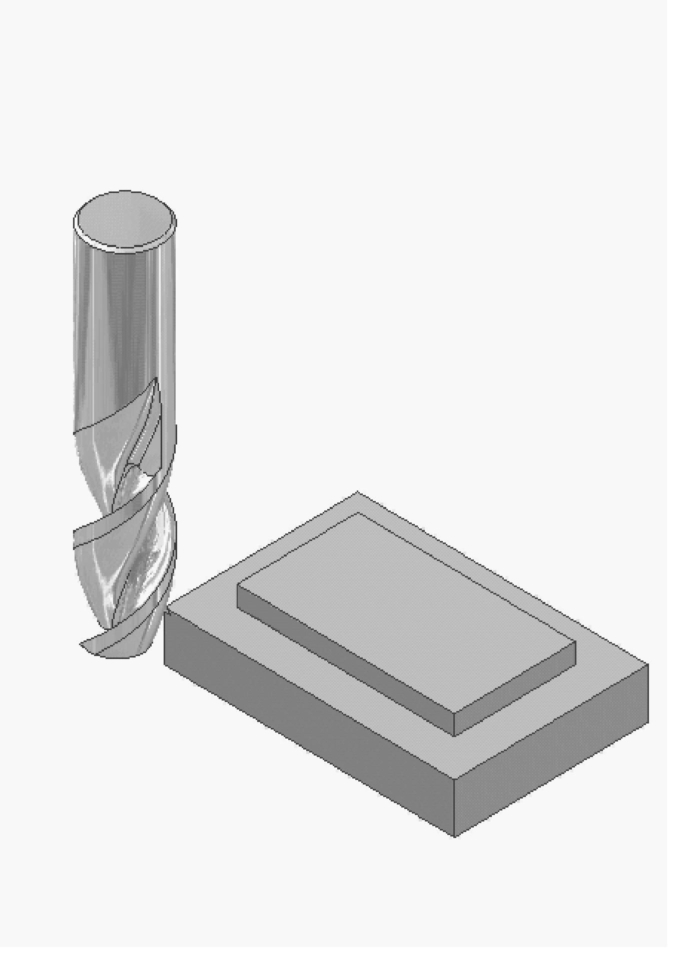
The Program

Program Codes	Action
%	Program Start
O999	
G20 G40 G49 G80 G99	
G91 G28 Z0	
G90	
T1M6	Load tool 1, ¾” HSS endmill.
S3056 M3	Set the spindle RPM and direction.
G0 G90 G54 X-0.01 Y-0.4	Go to initial position in the WCS using fixture offset G54.
G43 H1 Z-0.2 M8	Rapid to depth with length compensation, coolant on.
G1 Y2.01 F24.	Feed machining the left step.
X3.01	Machine top step.
Y-0.01	Machine right step.
X-0.4	Machine lower step and feed off the part.
G0 Y-0.4	Position move.
G41 X0.375 D1	Move to turn compensation on. Note the D code.
G1 Y1.625	Left edge.
X2.625	Top edge.
Y0.375	Right edge.
X-0.4	Lower edge. Since G40 is in the next block, center of the cutter ends at X-0.4
G0 G40 Y-0.4	Turn compensation off. Center of cutter now at X-0.4 Y-0.4
Z0.1	Lift above the part.
M9	Program end.
M5	
G49	
G91 G28 Z0	
G28 Y0	
G90	
M30	
%	

Program Animation

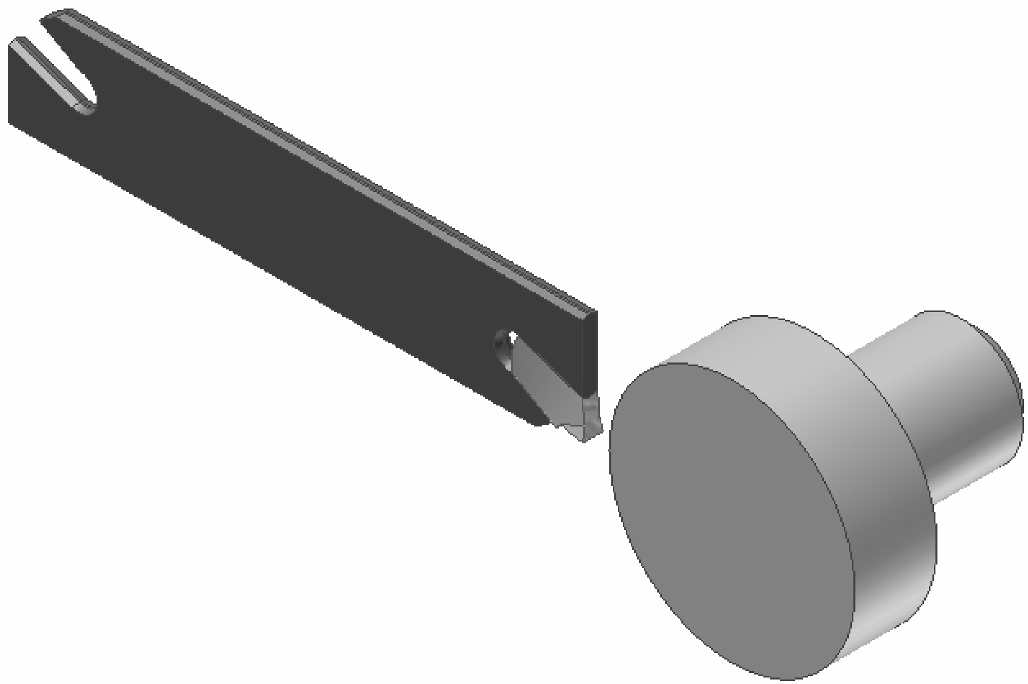
Select this link to
start the animation.

The animation starts with the cutter in its initial XY location and Z depth after the G43 line. The animation ends with the G0 Z0.1



CNC Applications

Parting on Turning Centers

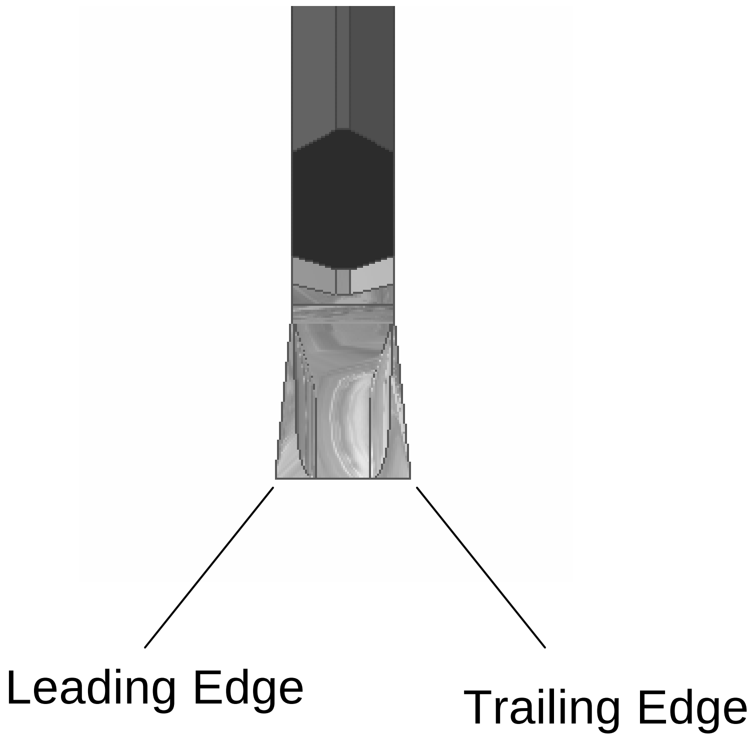


The Parting Operation

- Parting and grooving are very similar except parting removes the part from the end of a bar while grooving adds a groove to the part's profile.
- Parting of small parts can be done with a G1 to feed in and a G0 to rapid out.
- Larger parts should use the G75 parting cycle which pecks the cut so the chips break up.
- Be careful of parting parts completely off unless the machine has a parts catcher. Flying parts can damage tooling!

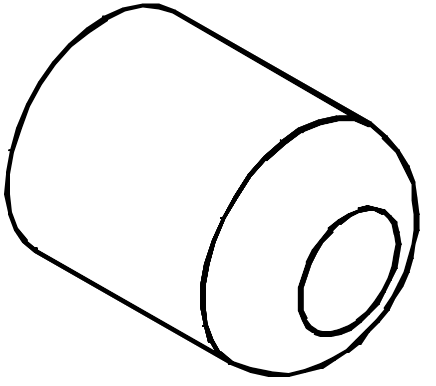
Setting the Parting tool

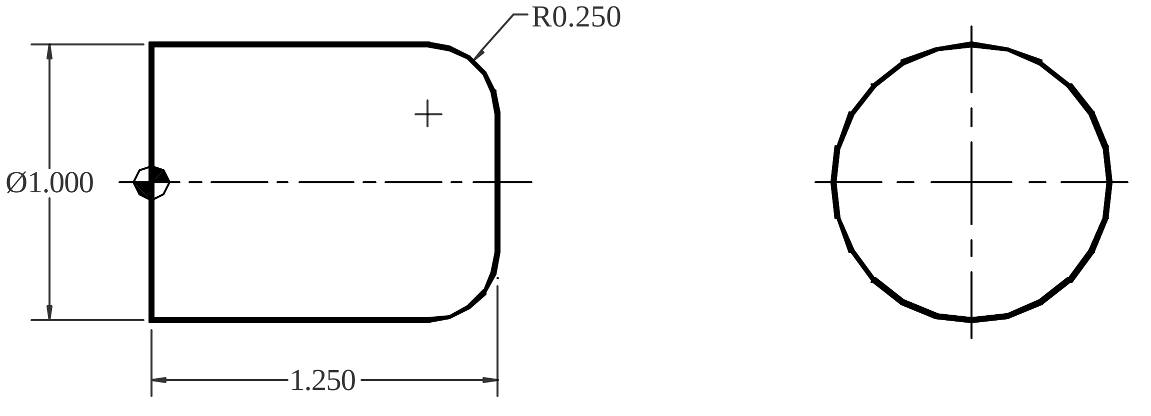
When programming and setting up a parting tool, you must decide where the tool offset will be taken from – either the leading or trailing edge. Then, program accordingly. In our examples, we set the parting tool offset on the leading edge of a 0.125” wide insert, so our Z value is Z-0.125 when cutting the part off to the origin.



A Simple Parting Example

We will assume all turning is done on this part, and we will just part it off with T05 at 600fpm and 0.004ipr. We'll stop at X0.050 to prevent the part from flying off the bar since our machine does not have a parts catcher. The operator will then have to wiggle the part to break it off.





Parting

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0505	Load the parting tool.
G54	
G50 S4000	Cap the RPM.
G96 S600 M3	Set the cutting speed to 600fpm, forward direction.
G0 Z-0.125	Rapid to the starting point for parting in Z
X1.1	and in X.
G1 X0.050 F0.004	Feed in.
G0 X1.1	Rapid out to the initial point.
M9	Program end.
M5	
G28 U0	
G28 W0	
M30	
%	

The G75 Parting Cycle

Small parts may be parted off by simply feeding the cutter straight into the part and then rapiding away, but larger parts require the G75 peck cycle to break the chips up and prevent them from clogging the cutter. The format is as follows:

```
G75 Xendx Qpeckdepth Ffeed
```

Where `endx` = X diameter at the bottom, generally 0 or slightly more than 0
`peckdepth` = how much to advance at a time
`feed` = feed rate for the parting cycle

Like all cycles, you must position the cutter at the cycle start point using G0 blocks. Then, at the end of the cycle, the cutter will return to its cycle start point.

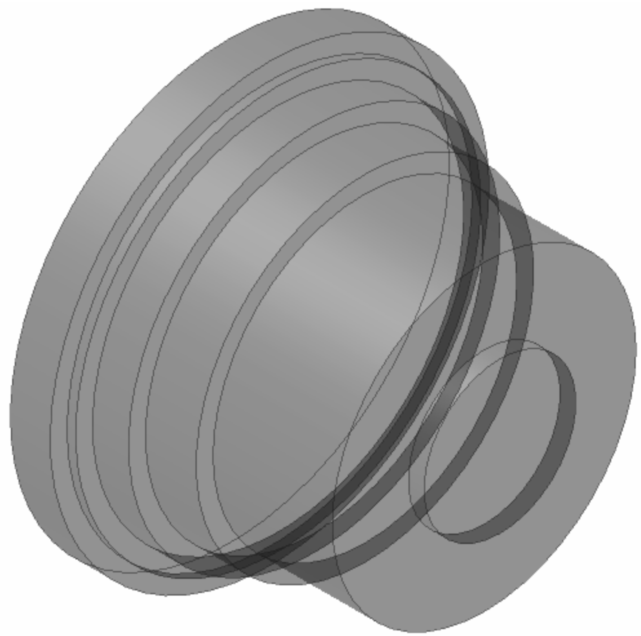
The next program shows the parting cycle being used to cut off our same example part.

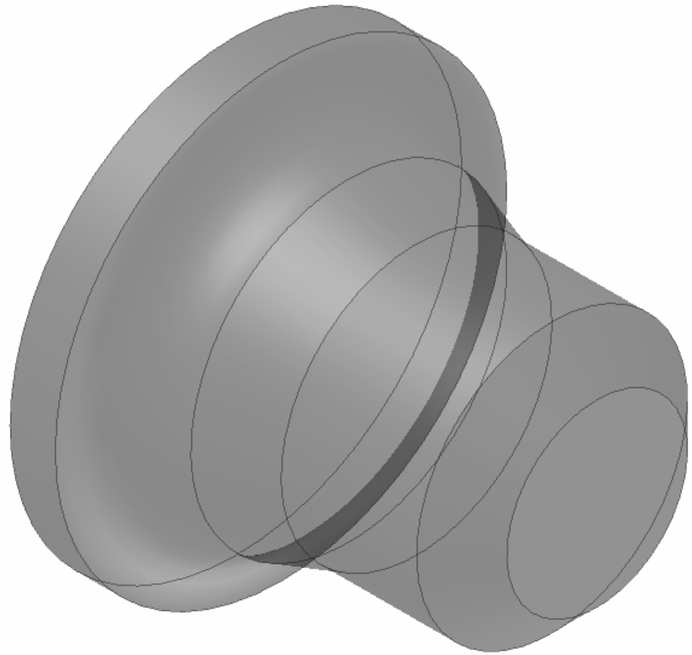
Parting with G75

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0505	Load the parting tool.
G54	
G50 S4000	Cap the RPM.
G96 S600 M3	Set the cutting speed to 600fpm, forward direction.
G0 Z-0.125	Rapid to the cycle starting point for parting in Z
X1.1	and in X.
G75 X0.050 Q0.25 F0.004	Parting peck cycle with 0.25” pecks. Returns to X1.1
M9	Program end.
M5	
G28 U0	
G28 W0	
M30	
%	

CNC Applications

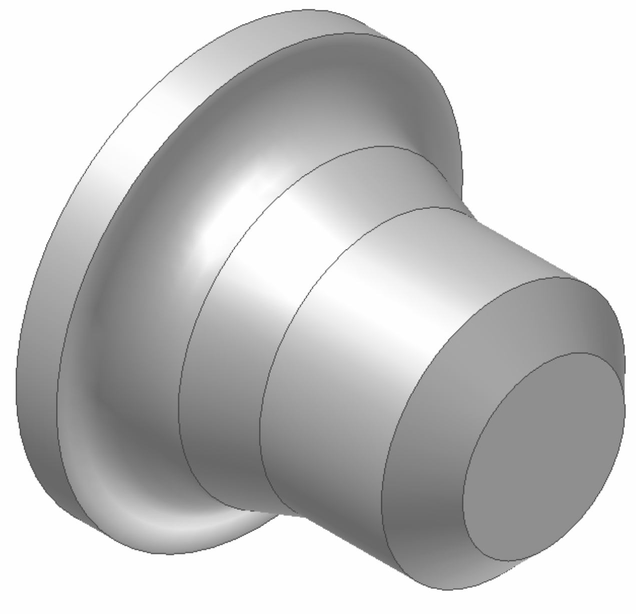
Roughing and Finishing Cycles for Turning Centers





The Problem

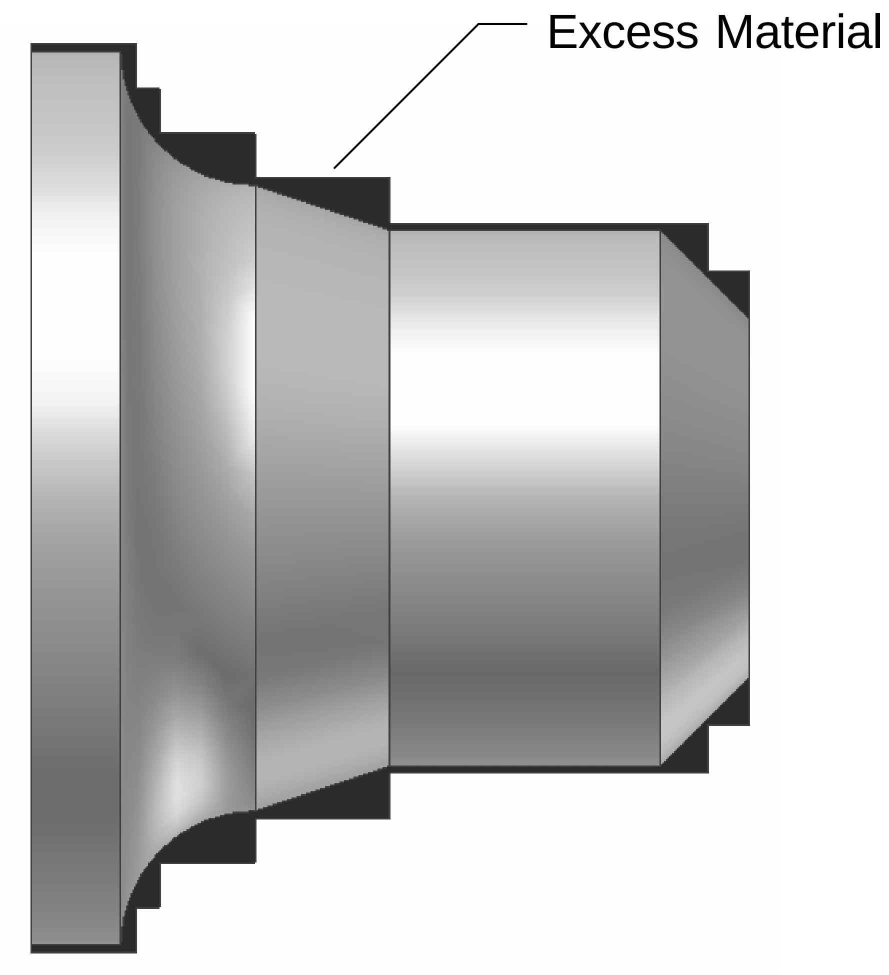
In turning, we frequently encounter parts similar to the examples we have been using with multiple diameters, tapers, chamfers, and radii. These features pose problems for roughing.



Roughing with G90

If we use the G90 rectangular turning cycle to rough the part, the excess material remains as shown with varying amounts of material in different locations.

This presents problems for the finishing cutter since the non-uniform depth of cut does not give predictable results when finishing.

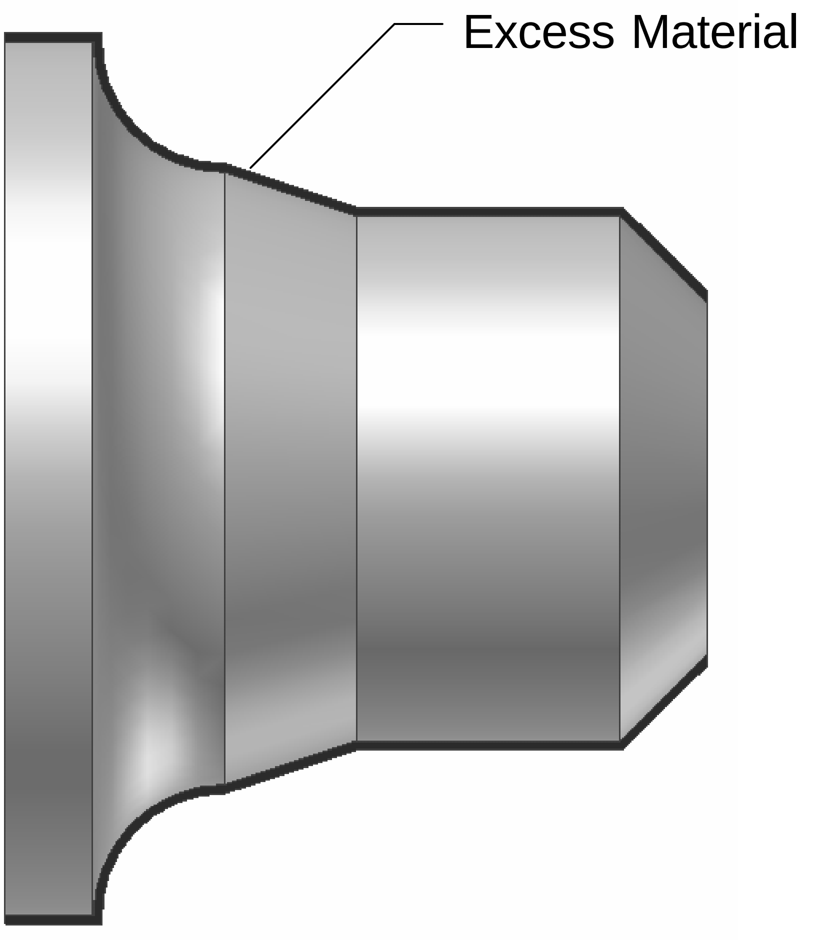


What We Really Want

What we really want is a roughing cycle that leaves a uniform amount of material so the finishing cutter will perform its job properly.

Since this is such a common occurrence in turning, the control manufacturers use G71 to rough leaving a specified amount of excess material and G70 to finish.

These two cycles greatly simplify programming complex parts.



Program Format with G71 and G70

Load roughing tool and locate at initial point.

Program lines between N10 and N20 describe the FINISH pass only. The machine roughs for us.

The Finishing Cycle refers back to the same program lines that the roughing cycle used since those lines describe the finish pass.

```
.
G71 P10 Q20 U0.06 W0.005 D1250 F0.012 S600
N10 G0 …
.
.
N20 G0 G40 …
```

Tool Change – Load the Finishing Tool, Turn Comp On

```
.
G70 P10 Q20
.
```

Your program loads the roughing tool and locates it at the cycle start point. The feed and speed values in the G71 line are used for roughing. Any speed and feed values in the N10-N20 program lines are used for finishing. The machine roughs based on the finish pass data. Then, load the finish tool and the G70 uses the same N10-N20 lines to cut the finish pass.

The G71 Roughing Cycle

You can probably tell the G71 format from the previous slide, but we'll give more explanation here:

```
G71 Pstartn Qendn Ufinishx Wfinishz Ddeltax Froughf Sroughs
```

Where `startn` = starting sequence number
`endn` = ending sequence number
`finishx` = material to be left on diameters (diameter)
`finishz` = material to be left on faces
`deltax` = integer value for radial depth of cut
`roughf` = feed rate to be used while roughing
`roughs` = spindle RPM or CSS (depending on G96 or G97) to be used while roughing

The machine advances by D depth of cut and machines close to the finish size. The values of U and W determine how close the machine comes to the finish size. When the G71 has completed, the part looks just like the finished part except it is oversize by the U and W values (bear in mind we are roughing here, so the surface finish will probably be rough as well).

The G70 Finishing Cycle

The format for the G70 finishing cycle is much simpler than for the G71 roughing cycle:

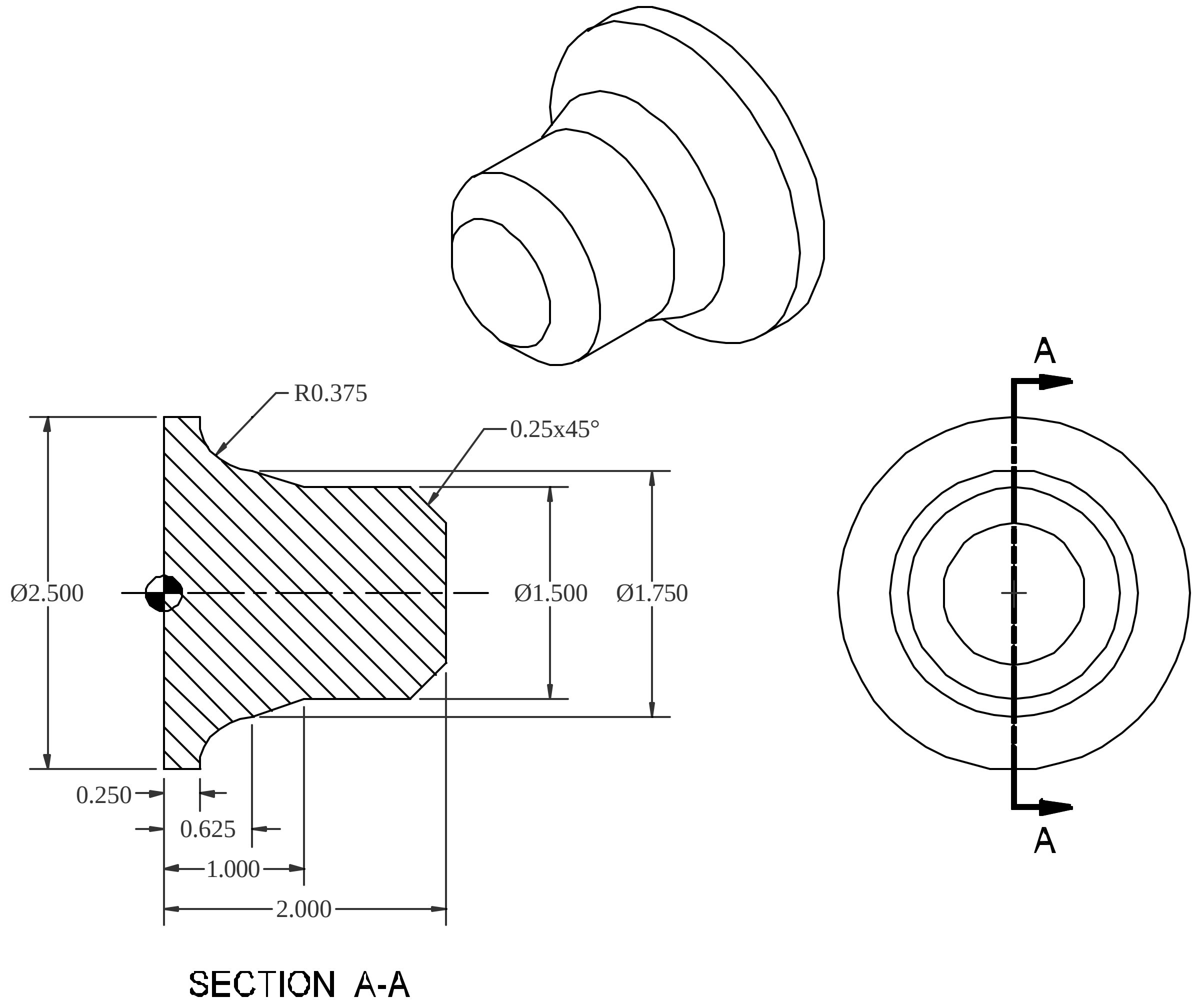
```
G70 Pstartn Qendn
```

Where `startn` = starting sequence number
endn = ending sequence number

You must load the finishing tool and then position the cutter 0.2” in X and 0.1” in Z away from the roughing cutter’s initial cycle start point. Then, move to the same start point turning on tool nose radius compensation. Program the G70. Your desired finishing speeds and feeds should be programmed in the N10-N20 blocks. G71 ignores these, only G70 uses them. The last line of the finish pass, the N20 block, should turn off tool nose radius compensation with a G40. Note that this has no effect on the G71.

A G71/G70 Example

Note that this is the same example we did for tool nose radius compensation. However, in this program we will rough the part at 600fpm and 0.012ipr feed with a C insert tool T02. Then we will finish the profile at 800fpm and 0.006ipr with a V insert T03. Finally, we will part the tool off with an 1/8” wide parting tool T05 at 600 fpm and 0.004ipr. We will use G41 nose radius compensation for the finish cut with T03 only.



Follow Planning and Programming Steps (1-5)

1. Examine drawing.
2. How will we hold the raw material – in a 3 jaw chuck.
3. Decide what cutters to use – given the following (use CSS for all cutters):
   - Roughing – C insert at 600fpm and 0.012ipr, T02
   - Finishing – V insert at 800fpm and 0.006ipr, T03
   - Parting – 1/8” wide parting tool at 600fpm and 0.004ipr, T05
4. Write down the exact sequence of operations:
   A. Face the part to length using T02.
   B. Rough the profile leaving 0.060” excess on diameters and 0.005” on faces.
   C. Finish the profile with cutter compensation.
   D. Part to X0.050 with the G75 parting cycle.
   E. Program end.
5. Convert the sequence of operations to a program:

Program Start
Face
Rough Turn
Finish Turn
Part
Program End

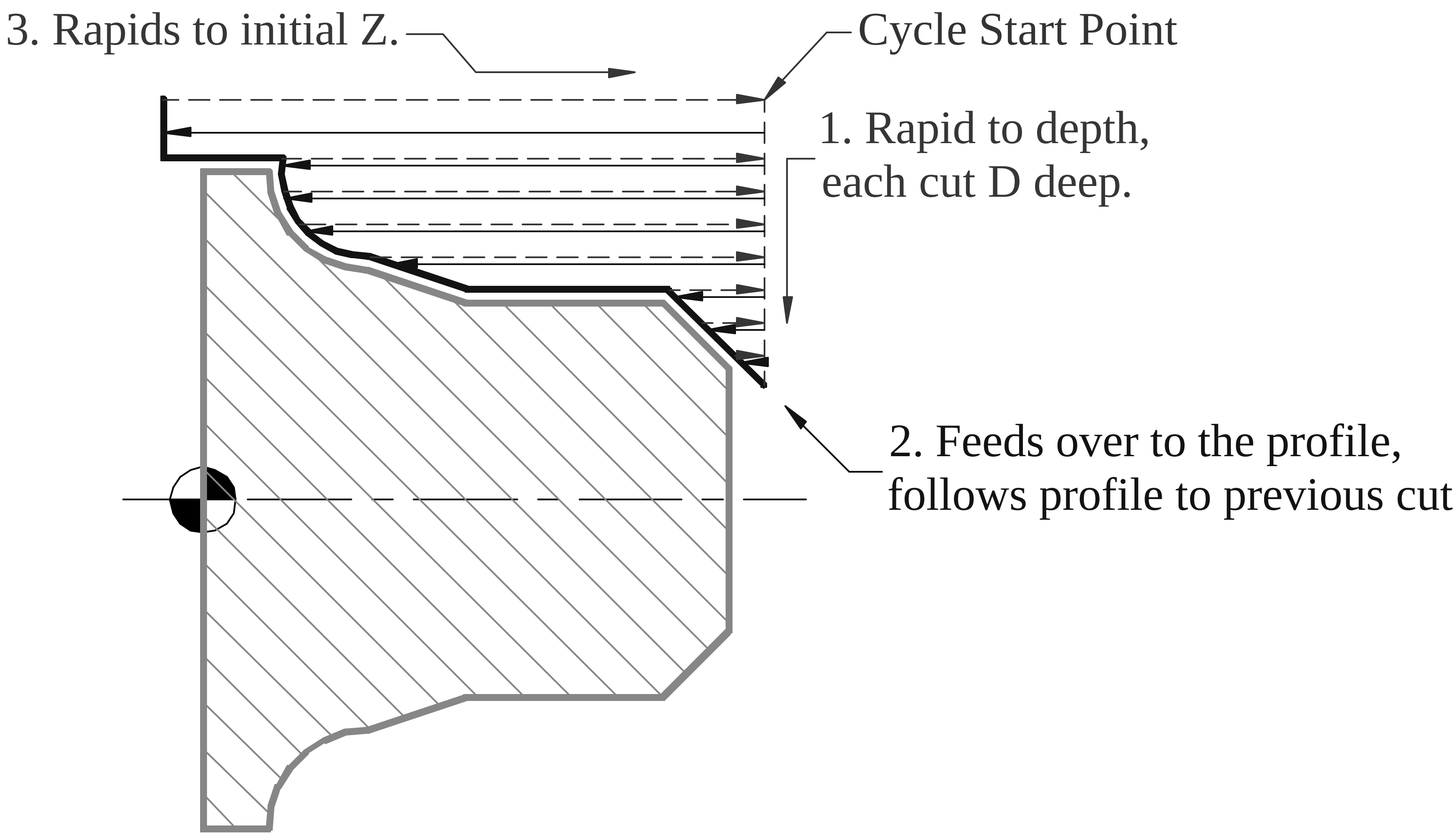
Facing

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0202	Load the C insert tool.
G54	
G50 S4000	Cap the RPM.
G96 S600 M3	Set the cutting speed to 600fpm, forward direction.
G0 Z2.005	Rapid to the starting point for facing in Z
X2.875	and in X.
G1 X0 F0.012	Rough Face
G0 Z2.1	
X2.875	Position for finish facing
Z2.00	
S800	Increase cutting speed for finishing, G96 is still active.
G1 X0 F0.006	Finish facing.
G0 Z2.1	Move to initial position for the roughing cycle in Z
X2.875	and in X.
.	.
.	Remainder of the program follows.
.	.

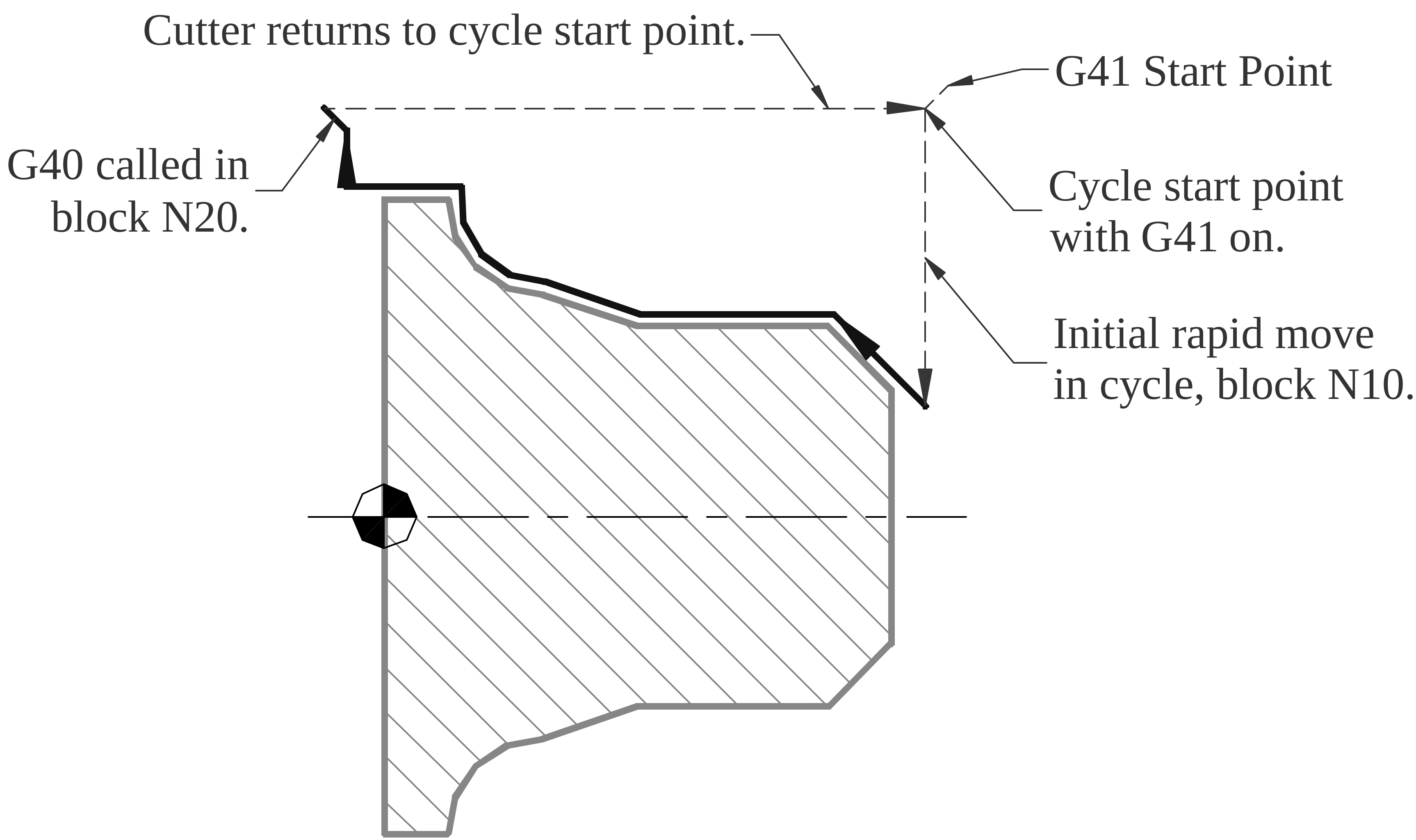
Roughing & Finishing

Program Codes	Action
G71 P10 Q20 U0.060 W0.005 D1250 S600 F0.012	Roughing cycle parameters.
N10 G0 X0.8	Move to the start of the chamfer, 0.1 clear in Z.
S800	Set finishing cutting speed (G71 uses 600).
G1 X1.5 Z1.75 F0.006	Machine the chamfer.
Z1.0	Machine the straight 1.0” diameter.
X1.75 Z0.625	Machine the taper.
G2 X2.5 Z0.25 I0.375	Machine the radius.
Z-0.15	Feed clear in Z leaving room for the parting tool.
X2.875	Feed clear in X.
N20 G40 X3.075 Z-0.25	Move to turn off nose radius compensation.
M9	
M5	Tool change.
G28 U0	
G28 W0	
T0303	Load the V insert tool.
G54	
G50 S4000	
G96 S800 M3	
G0 Z2.2	Move to tool nose compensation point in Z
X3.075	and in X.
G41 X2.875 Z2.1	Turn on tool nose radius compensation.
G70 P10 Q20	Perform the finishing cycle.
.	
.	Remainder of the program follows.
.	

How the G71 Works



How the G70 Works



Parting and Program End

Program Codes	Action
G28 U0 G28 W0 T0505 G54 G50 S4000 G96 S600 M3 G0 Z-0.125 X2.6 M8 G75 X0.05 Q0.25 F0.004 M9 M5 G28 U0 G28 W0 M30 %	Tool change. Locate the parting tool in Z and in X. Part off in 0.25” increments. Program End

CNC Applications

Homework Assignment 1

Name ___________________

Find the spindle RPM for the following cases. Show all work including the table and page number where you found the information. Only neat and legible work will be graded.

1. Turning 1117 CD steel 200 HB, 1.260" diameter with tough, coated carbide cutter. 1/4" depth of cut and 0.014 ipr feed.

2. Turning annealed A-10 tool steel 225HB 5" diameter with a hard, coated carbide cutter. 0.100" depth of cut, 0.009 ipr feed.

3. Turning 1117 CD steel 200 HB, 0.750" diameter with a hard, coated carbide cutter. 0.060" depth of cut and 0.005 ipr feed.

Homework Assignment 2

Name ___________________

Find the spindle RPM and feed rate in IPM for the following cases. Show all work including the table and page number where you found the information. Only neat and legible work will be graded.

1. Milling CD wrought aluminum with a 3/4” diameter 2 flute HSS end mill with 0.25 depth of cut.

2. Milling C36000 CD free cutting brass 100 HB with a 3" diameter 24 tooth HSS slotting cutter. Use the fastest feed possible.

3. Milling 8620 steel 200 HB with a 1" diameter HSS endmill with 4 teeth at 1/8” depth of cut.

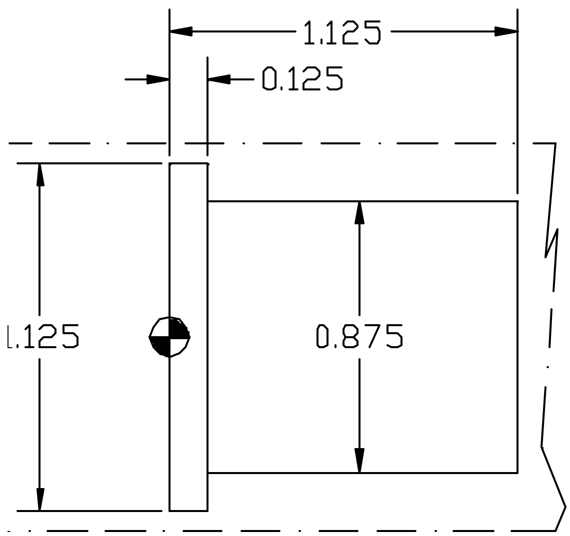
4. Milling annealed A-2 tool steel 225 HB with a 6" diameter HSS facemill with 8 teeth. Use the slowest feed possible.

CNC Applications

Turning Center Assignment 1

Name ____________________

Write a program for the CNC turning center to make the following part:



The part is made from 1117 steel at 200 HB with a rough bar size of 1.260". Make two facing cuts at 1/16" depth each and turn the part with T0202, and then part of the part with T0505. When parting off, stop at X0.050 so the part does not fly off.

Turn in the following items:

1. A printout of the program with an explanation of each program line.
2. A drawing with plotted points for each move made in the program. Reference these points to the program.

3. All speed and feed calculations.

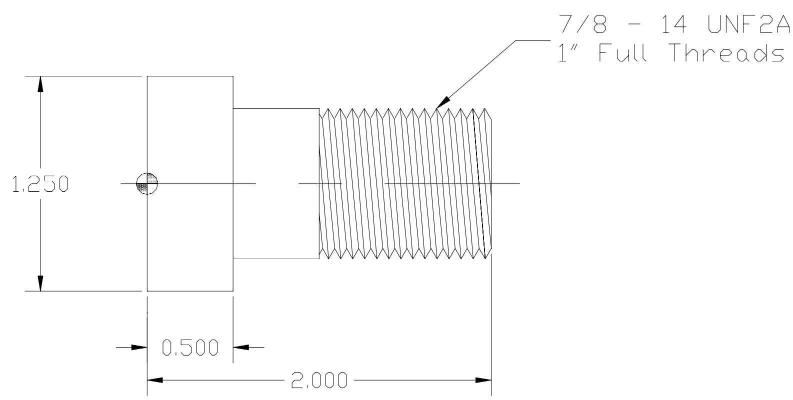
CNC Applications

Turning Center Assignment 2

Name ___________________

Write a program for the turning center to make the following part:

CNC Turning Lathe Project 2



The part is made from 1117 steel at 200 HB with a rough bar size of 1.260". With T0202, make two facing cuts at 1/16" depth each leaving 0.005 to finish, and then rough turn the part leaving 0.080 for finish on diameters and 0.005 on faces. Use G90 for roughing. Finish with T0303 and thread with T0404. When parting off with T0505, stop at X0.050 so the part does not fly off. Justify your speeds and feeds for each cut.

Turn in the following items:

1. A printout of the program with an explanation of each program line.

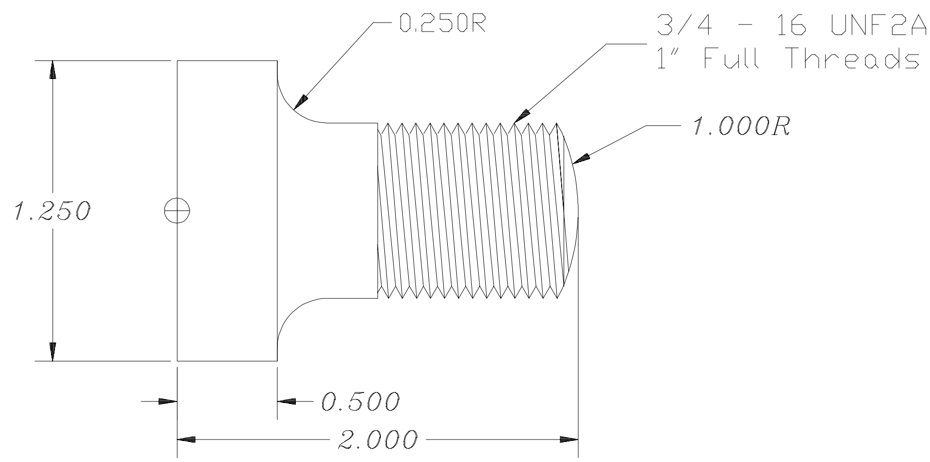
2. Speed and feed calculations.

CNC Applications

Turning Center Assignment 3

Name ___________________

Write a program for the turning center to make the following part:



The part is made from 1117 steel at 200 HB with a rough bar size of 1.260". With T0202, make two facing cuts at 1/16" depth each leaving 0.005 to finish, and then rough turn the part leaving 0.080 for finish on diameters and 0.005 on faces. Use G90 for roughing. Finish with T0303 using tool nose radius compensation, and thread with T0404. Cut the 1"R with tool 1 before threading. When parting off with T0505, stop at X0.050 so the part does not fly off. Justify your speeds and feeds for each cut.

Turn in the following items:

1. A printout of the program with an explanation of each program line.

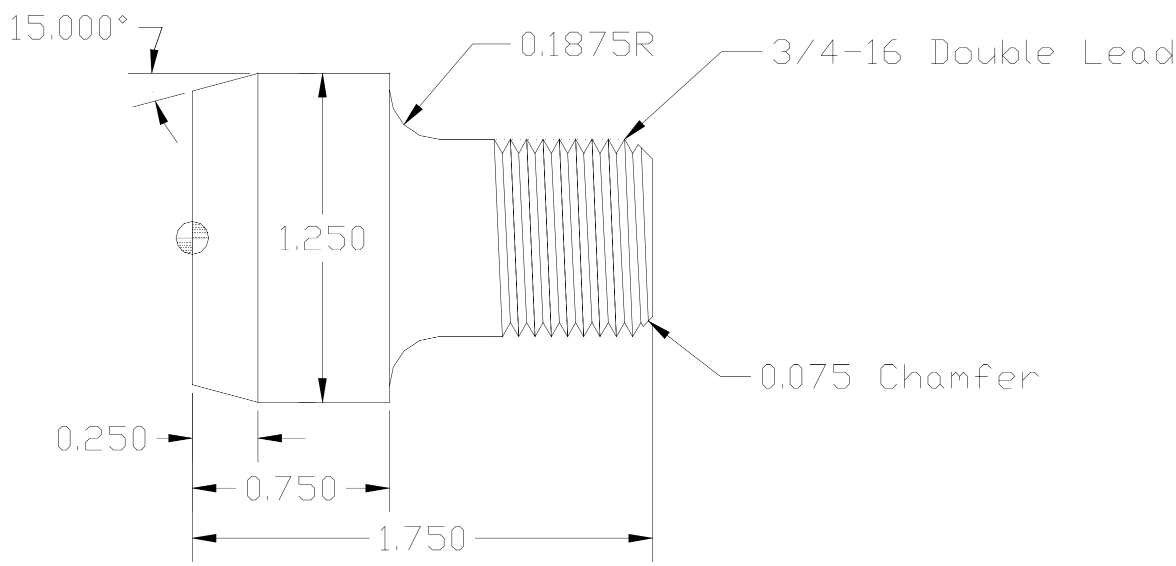
2. Speed and feed calculations.

CNC Applications

Turning Center Assignment 4

Name ___________________

Write a program for the turning center to make the following part:



The part is made from 1045 steel at 200 HB with a rough bar size of 1.262". With T0505, make two facing cuts at 1/16" depth each leaving 0.005 to finish, and then rough turn the part leaving 0.080 for finish on diameters and 0.005 on faces using G71. Be sure to use nose radius compensation. Use G70 for finishing with T0303. Thread with T0404. Use G75 for parting with T0505, and stop at X0.050 so the part does not fly off.

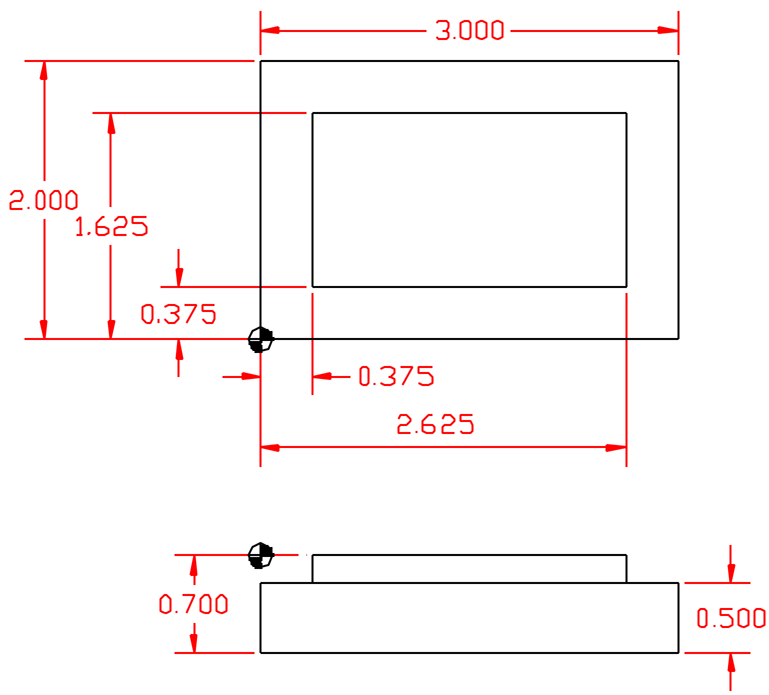
Turn in a printout of the program with an explanation of each program line.

CNC Applications

Milling Program Assignment 1

Name ____________________

Write a program for the Haas CNC machining center to make the following part:



The part is made from CD aluminum. Use a 3/4" endmill to rough the profile 0.010" oversize and then finish mill using the same cutter. Turn in the following items:

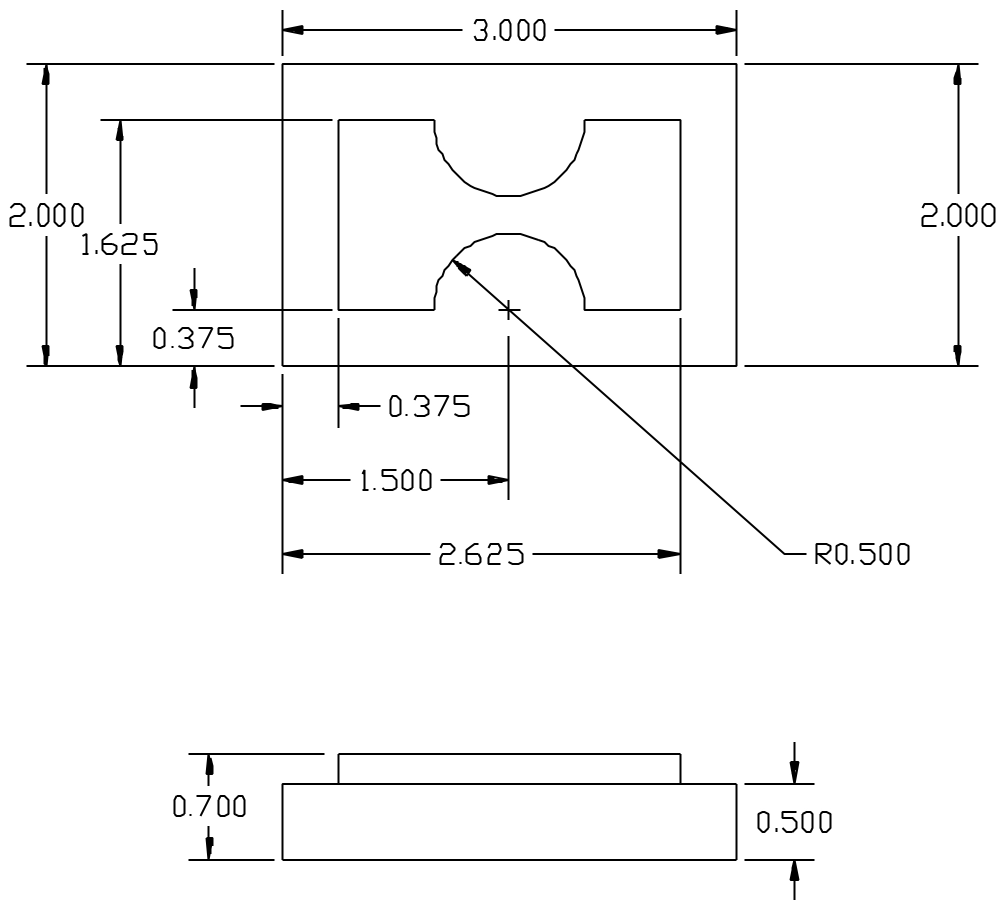
1. A printout of the program with an explanation of each program line.
2. All RPM and feed calculations.
3. A sketch of the program showing the center of the cutter at each direction change.

CNC Applications

Milling Program Assignment 2

Name ___________________

Write a program for the Haas CNC machining center to make the following part:



The part is made from CD aluminum. Use a 3/4" endmill to rough all features 0.010" oversize and then finish mill using the same cutter. Turn in the following items:

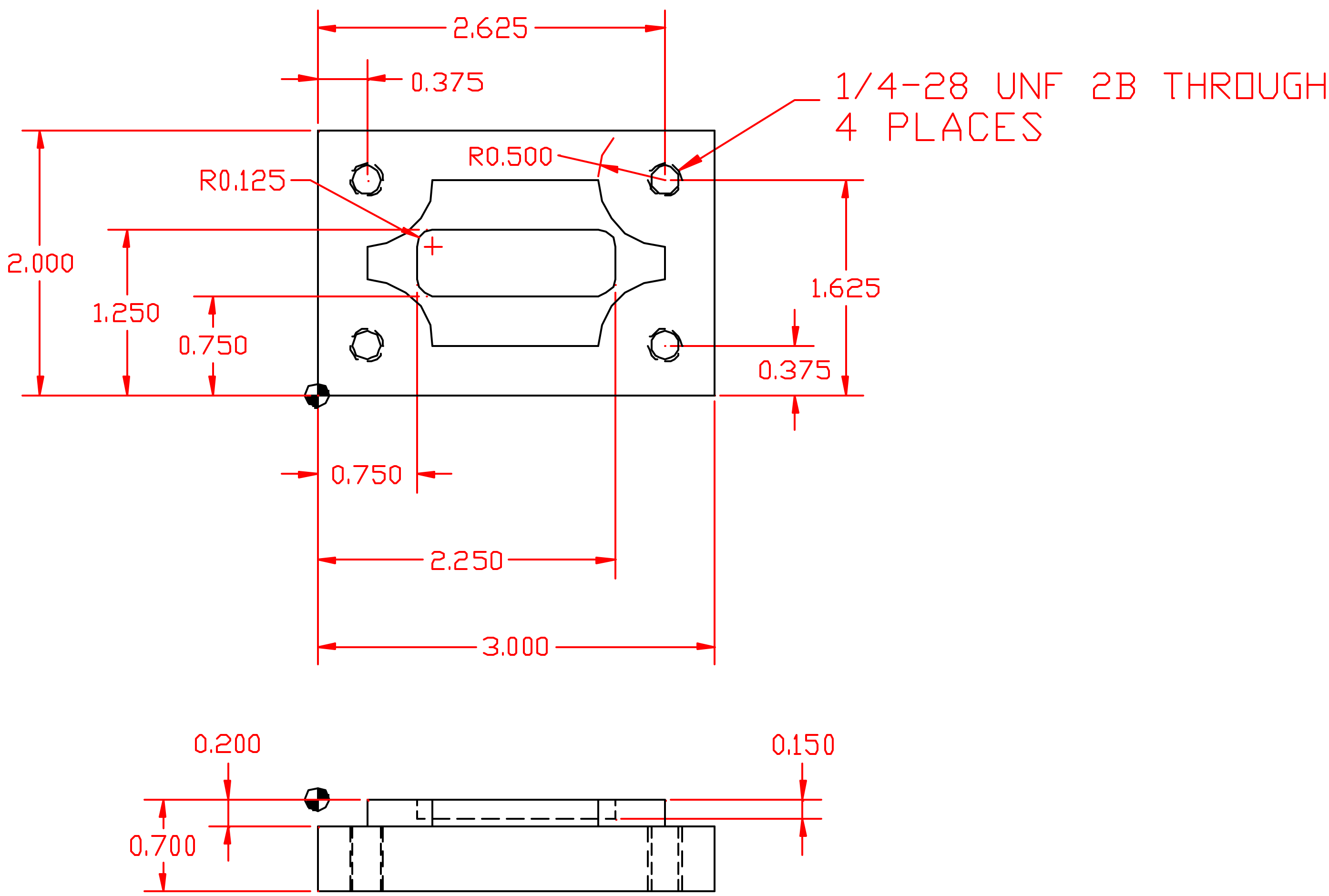
1. A printout of the program with an explanation of each program line.
2. All RPM and feed calculations.
3. A sketch of the program showing the center of the program at each direction change.

CNC Applications

Machining Center Program Assignments 3 & 4

Name ____________________

Write two programs for the Haas machining center to make the following part. Program 3 machines all features except the holes with a ¾" endmill (T1) and a ¼" endmill (T17). Program 4 centerdrills (T20), drills (T19), and taps (T18) only.



Turn in the following items:

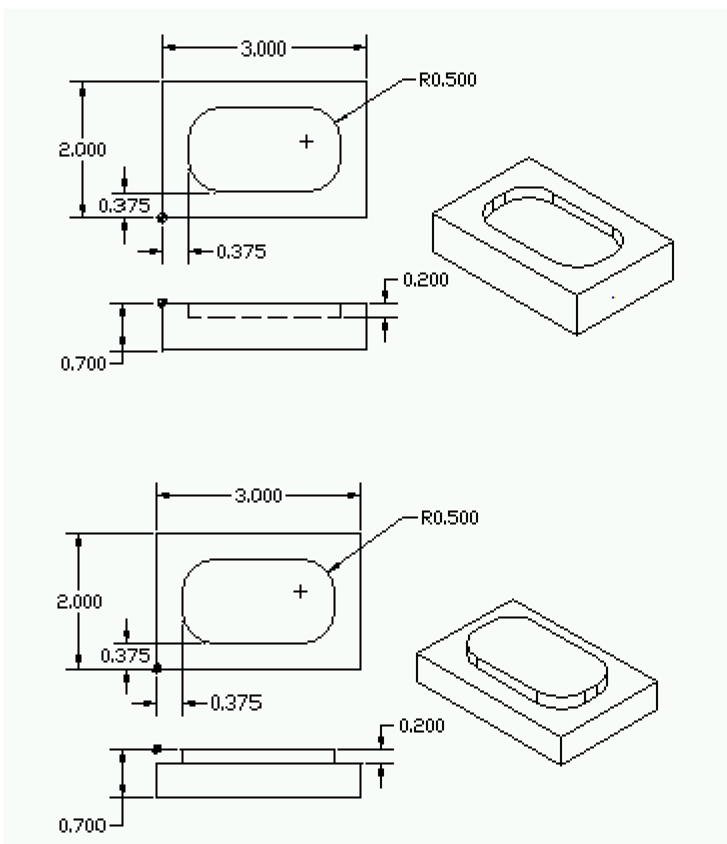
1. All speed and feed calculations.
2. A printout of the programs.

CNC Applications

Machining Center Program Assignments 5 & 6

Name ________________

Write two programs for the Haas machining center to make the following parts. Program 5 machines the length and thickness of the female part with a ¾” endmill (T1) and mills the pocket with a ½” endmill (T2). Program 6 mills the length, thickness, and boss of the male part with a ¾” endmill (T1). Use G41 cutter diameter compensation for the pocket and boss. The two parts should fit together snugly when finished.



Turn in the following items:

1. All speed and feed calculations.
2. A printout of the programs.