Haas Factory Outlet

A Division of Productivity Inc

Lathe Series Training Manual

Live Tool for Haas Lathe (including DS)

Created 020112-Rev 121012, Rev2-091014

This material is to be used as a guide to operation of the machine tool. The Operator is responsible for following Safety Procedures as outlined by their instructor or manufacturer's specifications.

NOTE: Downloading and/or other use of this manual does not certify completion of the Training Course. This manual is for reference only.

Live Tool for Lathe Training Manual
Table of Contents

For more information on Additional Training Opportunities or our Classroom Schedule,
Contact the Productivity Inc Applications Department in Minneapolis:

☏ 763.476.8600

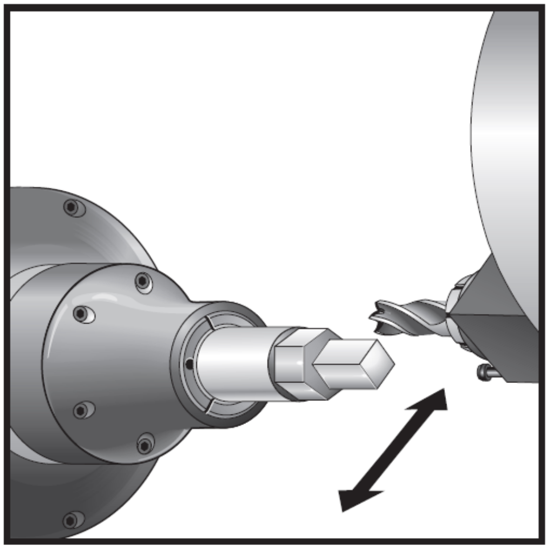
Visit us on the Web: www.productivity.com

Click on the Training Registration Button

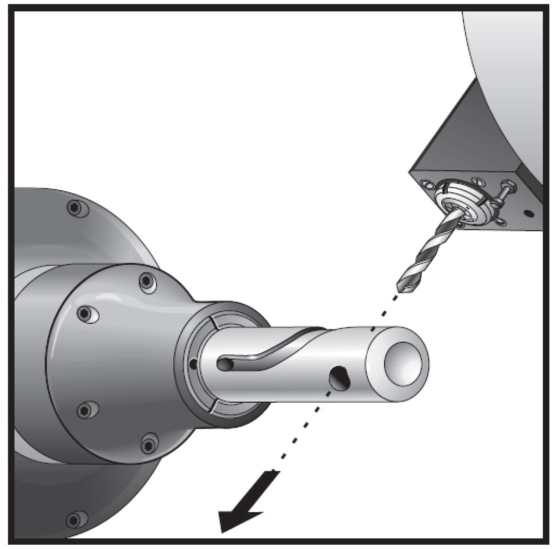
✉ trainingmn@productivity.com

Created 2/1/12 – CK/DF (Added DS Section II 5/1/12); Rev 12/10/12, Rev2-9/10/14 CK

Live Tool for Haas Lathes Training Introduction


Axial Tool

AXIAL (Face-Working) Tool


Radial Tool

RADIAL (Cross-Working) Tool

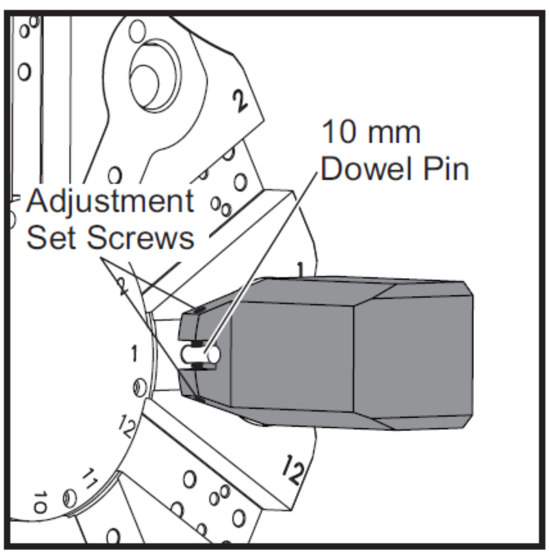
The Live Tooling option allows driven VDI axial and radial tools to perform secondary operations. The Haas standard turret is configured with Haas VDI adapters. These adapters locate, orient and drive standard 40mm VDI tools. Also a VDI turret is available with live tooling option. The VDI turret is designed to fit standard 40mm shank VDI tools.

Mounting Radial (Cross) Live Tool Holders on Turret and Alignment

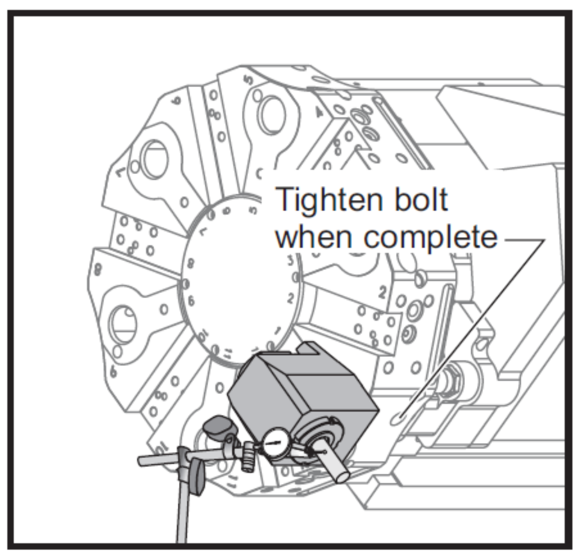
Radial live tool holders should be adjusted for optimum performance during milling with the Y-Axis. The body of the tool holder can be rotated in the tool pocket relative to the X-axis. This allows for adjustment of the parallelism of the cutting tool with the X-axis. Adjustment set -screws are standard on all radial live tool heads. A 10mm dowel pin is required for alignment. For ease of removal 10mmx30mm ground pins with air relief and ¼-20 holes are recommended. These can be purchased thru McMaster Carr.

Radial Live Tool Mounting and Alignment

1) Install 10mm Ground Pin on the turret.
2) Before mounting live tool remove and thoroughly clean VDI bolt. Also clean the inside of the VDI holder on the turret.
3) Apply a thin film of way lube on the internal contact surfaces of the Radial Live Tool.
4) Mount Radial Live Tool and snug adjustment set screws against the dowel pin at a visually-even and centered position.
5) Snug the VDI Allen bolt to allow for some movement and adjustment of the tool. Ensure the back face of the tool holder is flush with the face of the turret.
6) Position the Y-axis at zero.



Adjust Alignment with Set Screws



Indicate Dowel Pin or Gauge Pin

7) Install a dowel or gauge pin on the holder like you would install the cutting tool.
8) Make sure the tool sticks out at least 1.25" (32mm). This will be used to run the indicator across it to insure parallelism to X-axis.
9) Set an indicator with a magnetic base on a rigid surface (for example, the tailstock base).
10) Position the indicating tip on the end point of the pin and zero the indicator dial.
11) Sweep the indicator along the pin to measure the parallelism between the pin and X-axis.
12) Adjust set screws mentioned on # 3 and keep indicating across the top of pin until tool is properly aligned and parallel to X-axis.
13) Tighten VDI Allen bolt to recommended torque.
14) Repeat steps # 1 to # 8 for every radial tool used in set-up.

Mounting Axial (Face) Live Tool Holders on Turret and Alignment

1) Before mounting live tool remove and thoroughly clean VDI bolt. Also clean the inside of the VDI cavity on the turret.
2) Apply a thin film of way oil on the internal contact surfaces of the Axial Live Tool.
3) Mount Axial Live Tool aligning on alignment pin. Snug the VDI Allen bolt. Ensure the back face of the tool holder is flush with the face of the turret.
4) Tighten VDI Allen bolt to recommended torque. For VDI 40 tooling, the recommended torque is 35 to 45 ft-lb (Global CNC Industries).

Installation of Tools in Live Tooling Holders

1) Insert the tool bit into an ER-32 collet. Thread the ER-32 collet nut insert into the collet housing over tool bit and ER-32 collet
2) Place spanner wrench over the pin of the live tool holder and lock it against the collet
3) Engage the teeth of the collet wrench and tighten

Setting Live Tool Offsets

Touching-Off Radial Live tools

When touching off radial live tools use the following procedure. Example: If using a ½" (12mm) diameter end mill, add ¼" (6mm) to the Z offset for that tool. The added value MUST be negative (radial tools only.)

1) Press the HANDLE JOG key.
2) Press .1/100. (The lathe will move at a fast rate when the handle is turned).
3) Toggle between the X and Z jog keys until the tool is close to the side of the part.
4) Touch off of the X-axis offset manually or with a tool presetter in the same manner as any other tool on the turret.
5) Touch off the Z-axis on the face of the part or with a presetter in the same manner as any other tool in the turret. It is necessary to add the **negative** value of the radius of the tool to Z axis column of the respective tool in the tool geometry register. The new value will make the center of the tool coincide with the face of the part.

Touching-Off Axial or Face Working Live Tools

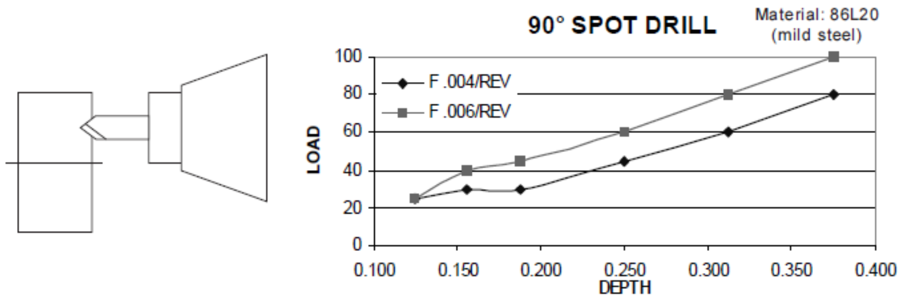
1) Press the HANDLE JOG key.
2) Press .1/100. (The lathe will move at a fast rate when the handle is turned).
3) Toggle between the X and Z jog keys until the tool is close to the side of the part.
4) Touch off of the X-axis offset manually or with a tool presetter in the same manner as any other tool on the turret. Then you must (minus the X value) by the diameter of the tool that you are touching off. Alternately in jog mode cursor to the tool geometry x- value and depress F2. This value was determined at the Haas plant.
5) Touch off the Z-axis on the face of the part or with a presetter in the same manner as any other tool in the turret.

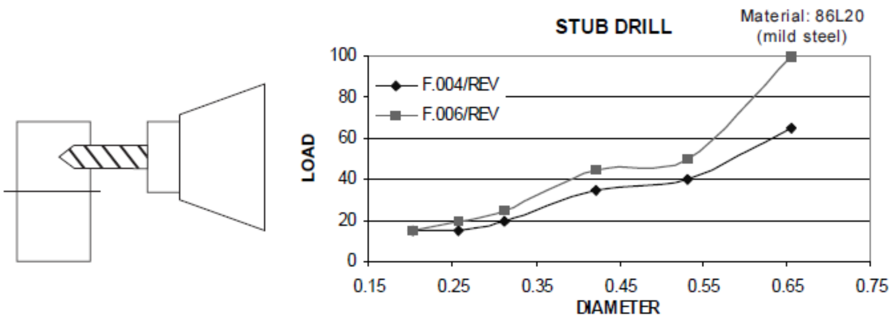
Live Tool Cutting Capacity

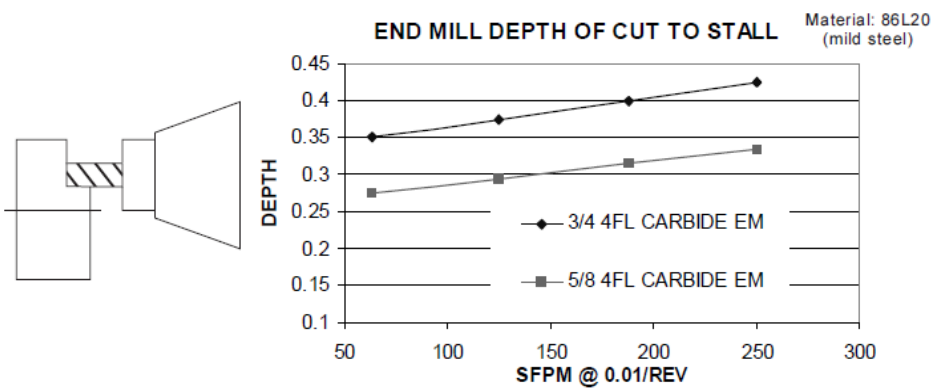
Haas live tooling is designed for medium duty milling, e.g.: 3/4" diameter end mill in mild steel max. The graphs on page 6 show spindle loads and depth of cut to spindle stall with various drill and mill diameters, feeds and speeds. Maximum live tooling drive speed is 3000 RPM. Live tooling is driven by a 5hp or optional 7hp motor.

The graphs on page 7 give torque information at various speeds and information on tool holder bearing life.

HAAS SL LIVE TOOLING CUTTING CAPACITY

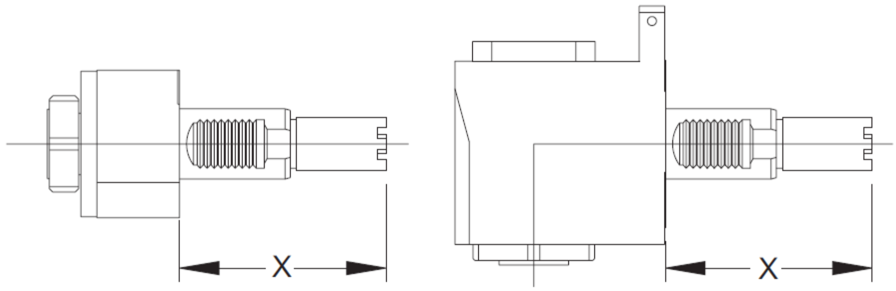






ES-0125B 1/99

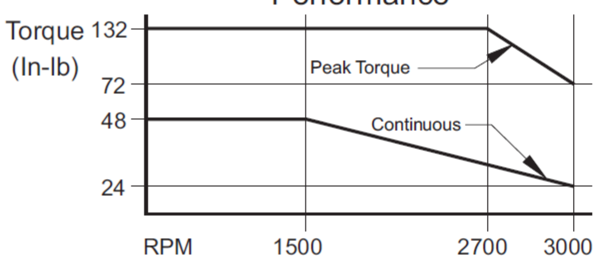
Live Tooling Tool Holders and Torque Chart



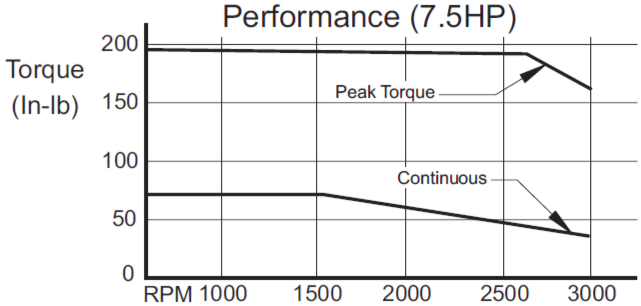
	in/(mm)	HAAS TURRET * SL-20/30	HAAS TURRET * SL-40	VDI TURRET SL-20/30	VDI TURRET SL-40/TL-15
AXIAL TOOLHOLDER	X-DIM P/N	4.628 (117.55) P/N 59-0082	5.128 (130.25) P/N 59-6352	4.128 (104.85) P/N 59-0083	4.628 (117.55) P/N 59-0082
RADIAL TOOLHOLDER	X-DIM P/N	4.628 (117.55) P/N 59-0084	5.128 (130.25) P/N 59-6353	4.128 (104.85) P/N 59-0085	4.628 (117.55) P/N 59-0084

* VDI ADAPTER KIT (HAAS P/N VDIA) REQUIRED.

Standard Spindle Drive Performance



Optional Spindle Drive Performance (7.5HP)



- Haas live tooling is designed for medium duty milling, e.g.: 3/4" diameter end mill in mild steel max.
- Large tool diameters may require reduction toolholders.
- Tool length should be kept to a minimum for toolholder rigidity.
- Typical life of toolholder bearing is 1000 hours with medium duty operation.
- Worn toolholder bearings will not machine properly.

Live Tooling Parameters

Parameter 72 LIVE TOOL CHNG DLAY

This parameter specifies the amount of time (in milliseconds) to wait after commanding the Live Tooling Drive motor to turn at the velocity specified by parameter 143. This process is required to engage the castle gear between live tooling motor and tool. It is only performed prior to the first M133 or M134 after a tool change. The value set from the factory is **500**.

Parameter 143 LIVE TOOL CHNG VEL

This parameter specifies the velocity to command the Live Tooling Drive motor for the period specified by parameter 72. This process is required to engage the castle gear between the live tooling motor and tool and is only performed prior to the first M133 or M134 after a tool change. The value set from the factory is **100**.

Parameter 278 LIVE TOOLING Bit 24

This is a new feature. For lathes fitted with the Live Tooling drive, this bit must be set to **1**. For all other lathes this bit must be set to 0.

Parameter 304 SPINDLE BRAKE DELAY

This parameter specifies the amount of time (in milliseconds) to wait for the main spindle brake to unclamp when spindle speed has been commanded, and also the amount of time to wait after the main spindle has been commanded to stop before clamping it. The value set from the factory is **500**.

Parameter 315 Bit 1 NO SPINDLE CAN CYCLE

This parameter bit must be set from 0 to 1 when using live tooling face drilling can cycles G81, G82, G83. It also must be changed if you are using G95 live tool face tapping cycle.

Live Tooling Code

Notes on Live Tooling

1) The live tool spindle will automatically turn itself off when a tool change is commanded.
2) The main spindle can be clamped (M14 and M15) for using the live tooling. It will automatically unclamp when a new main spindle speed is commanded or RESET is pressed.
3) Maximum live tooling drive speed is 3000 rpm.

G98 versus **G99**: G99 (feed per spindle revolution) is the default on a lathe. With most live tooling code **G98 (feed per minute)** is used as the spindle is not rotating at high rpm. The units are inches per minute or degrees per minute.

M133 (Live Tool Drive Forward)
Turns on the live tooling motor to a (PXXXX)rpm. Maximum speed is 3000 rpm. When the live tooling is engaged the live tool motor turns slowly for 500 milliseconds to engage the castle gear. This is set at the factory in the parameters. M133 P1000 turns on live tooling forward to 1000 rpm

M134 Live Tool Drive Reverse

M135 Live Tool Drive Stop

M14 Clamp Main Spindle
M14 clamps or turns on the spindle brake.

M15 Unclamp Main Spindle
M15 unclamps or turns off the spindle brake. The spindle will automatically unclamp when a tool change is commanded or when a new spindle angle is commanded. Also when RESET is pressed the spindle will unclamp.

M19 Orient Spindle (Optional)
M19 will orient the spindle to the zero position. A P or R value is used to orient the spindle to a specific position (in degrees.). Degrees of accuracy: P rounds to the nearest whole degree, and R rounds to the nearest hundredth of a degree (x.xx). The angle is viewed in the Current Commands Tool Load screen.

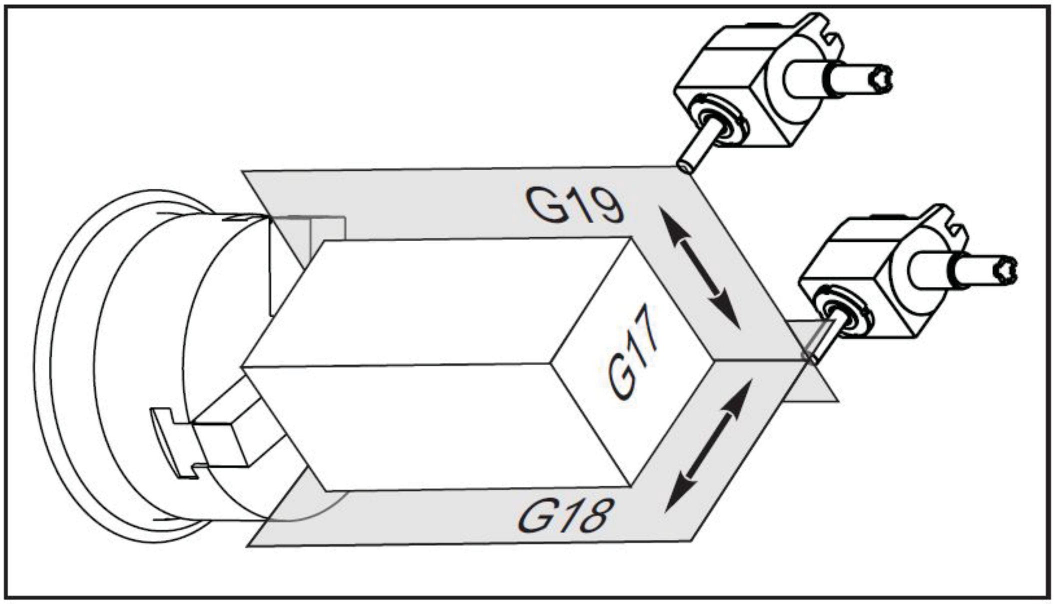
M119 will position the secondary spindle (DS lathes) the same way.

M154 C-Axis Engage

M155 C-Axis Disengage

These codes engage and turn on and off the C-axis motor. After engaging the C-Axis with M154, it is recommended that the following line **(G28 H0)** block be inserted. **H** is the incremental C-Axis command. **G28 H0** will take the machine home in the C axis (C0). This will ensure that the gears used for the C-Axis are fully engaged.

Plane Selection and Feed Rates for Live Tools



Above illustration shows the G17, G18, and G19 planes for throwing radii on a lathe. The default on a lathe is G18. This is used in normal turning operations to circulate interpolate radii on the OD or ID of the part. The radial tool shown above is moving along the Y axis. All radial or cross drilling can cycles need to be in G19 to work properly. With drilling operations the location is described by Y and Z and C axis. The depth of the hole called out by the X.   Also if a circular or rectangular pocket is being created on the OD using the Y and Z axis the machine must be in G19.

All axial or face working operations use G18 to properly work. With face or axial drilling cycles the location is described in the X and C axis. The depth prescribed by the Z axis.   The only face working operation not using G18 is the G112 Cartesian to Polar transformation which requires the machine to be in the G17 mode.

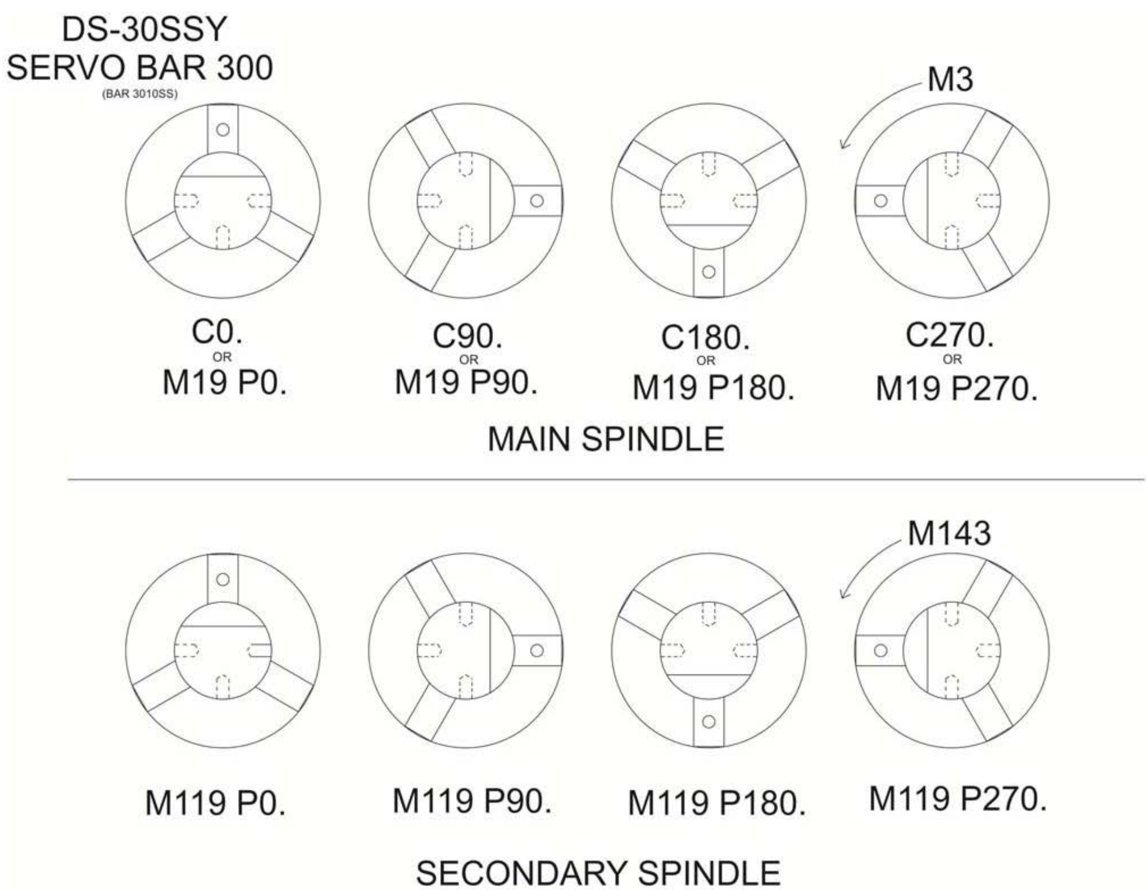
Illustrations above and next page taken from" Y-AXIS LATHE APPLICATIONS TRAINING", AP-100 Rev A Dec 2012 Haas Automation.

Spindle Orientation using M19 and the C-axis

M19 Orient Spindle (Optional)   M19 will orient the spindle to zero. Using a P or R value will orient the spindle to a specific position in degrees. Degrees of accuracy: P rounds to the nearest whole degree while R rounds to the nearest hundredth of a degree (X.XX). From figure below the direction of positive rotation will be clockwise from an operators view point facing the chuck.   A negative command will result in a counter clockwise rotation. Moves may be in incremental or absolute. See below figure.

M119 Orient Sub-Spindle   M119 will position the sub spindle on DS lathes in a similar fashion, a positive rotation will be seen from an operators view point as a clockwise rotation.
Note that during normal turning operations of M03 and M143 the spindles will be going in the counter clockwise direction.

M154 C-Axis Engage and **M155 C- Axis Disengage:** These codes engage and turn on and off the C-axis motor.



It is preferable to always use the C-axis on the main spindle as it is more accurate and repeatable. C-axis positioning is +/- .01 degree. The sub spindle must use M119 or G14 M19. If using the C-axis it is advisable to take the C-axis home first using **G28 H0.** This will insure the C-axis gears are fully engaged.

Different cycles also require specific feed rates.
Feed rates are generally in **G98 (in/min).** The only exceptions are tapping cycles which require **G99 (in/rev).** Generally all **axial** or face working cycles require **G18** (X-Z plane) except **G112** which must be in **G17** (X-Y Plane).
All cross working or **radial** cycles must be in **G19** (Y-Z plane). After using live tooling cycles return the machine to the default codes of **G18 (X-Z Plane)** and **G99 (in/min).**

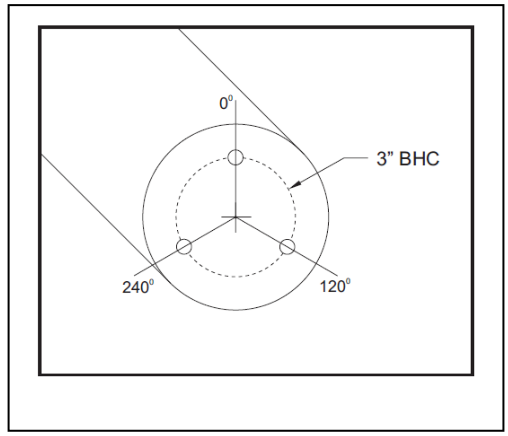
Plane Selection and Feed Rates for Different Canned Cycles

Canned Cycle	**Cycle Description**	**Plane Selection**	**Feed Rate**
Face Working Cycles			
G81	Drill	G18	G98
G82	Drill with Dwell	G18	G98
G83	Drill with Peck	G18	G98
G95,	Rigid Tap	G18	G99
G186	Rigid Tap Left	G18	G99
X-C Axis Milling	Manual Slots	G18	G98
X-Y Axis Milling	Manual Radius	G17	G98
G112	Cartesian to Polar	G17	G98
Cross Working Cycles			
G75	Drill with Peck	G19	G98
G241	Drill	G19	G98
G242	Drill with Dwell	G19	G98
G243	Drill with Peck	G19	G98
G195	Radial Rigid Tap	G19	G99
G196	Radial Rigid Tap Left	G19	G99

Axial or Face Machining with Live Tools

Program Example: Drilling using M19

Bolt Hole Circle 3 holes @ 120° on 3. Inch BHC




```
G0 X3.0 Z0.1
G98  (in/min feed)
M19 P0  (rotate spindle to 0 degrees)
M14 (clamp spindle)
M133 P2000 (turn on live tooling 2000 rpm)
G1 Z-0.5 F40.0
G0 Z0.1
M19 P120 (rotate spindle to 120 degrees)
G4 P3 (Dwell for servo stabilization)
M14 (clamp spindle)
G1 Z-0.5
G0 Z0.1
M19 P240 (rotate spindle to 240 degrees)
M14 (clamp spindle)
G1 Z-0.5
G0 Z0.1
M15 (unclamp spindle)
M135 (live tooling stop)
G53 X0
G53 Z0
M30
```

Using G81, G82, G83, G95, and G186 with live tooling

The above canned cycles above can be used with face or axial live tooling provided parameter 315 bit 1 (NO SPIN CAN) is set to 1. When this parameter is set to 1, the main spindle will not be activated during a canned cycle. If is set to 0 the canned cycle operates in the usual way by turning the main spindle on. The reference plane must be the X-Z plane, G18. They work the same way as described in the lathe manual except G98 In/min must be called up except in G95 and G186.

Same example of drilling 3” BHC using C-axis with G81 Code instead of M19

Bolt Hole Circle 3 holes @ 120° on 3” BHC

```
O0051
T101
G18
G54 G00 X3.0 Z0.1
G98
M154 (C-Axis engage)
C0.0
M133 P2000 (Live Tooling Drive Forward)
G81 Z-0.8 F40.0
C120.0
C240.0
G00 G80 Z0.1
M155 (C-Axis Disengage)
M135 (Live tool drive)
G53 X0
G53 Z0
M30
```

The same example using pecking cycle G83 with Q.

```
O0052
T101
G18
G54 G00 X3.0 Z0.1
G98
M154 (C-Axis engage)
C0.0
M133 P2000 (Live Tooling Drive Forward)
G83 Z-0.8 Q.2 F40.0
C120.0
C240.0
G00 G80 Z0.1
M155 (C-Axis Disengage)
M135 (Live tool drive)
G53 X0
G53 Z0
M30
```

Program Example: G95 Rigid Tapping

With G95 and G186 (reverse live tool rigid tapping) the **feed rate must be in G99 In/rev**. Also note below the live tooling is **not** turned on with **M133** but a **spindle speed S500** given instead. The spindle will automatically start by itself. Also note that the **G95 must be called** up with **each** new **location**, below it is a C value. After the can cycle is cancelled with G80 the live tool stop command M135 is placed on the next block.

<u>Tapping previous page holes.</u>

```
(LIVE TAP - AXIAL) (1/4 x 20 Tap)
T1111
G18
G99
M154 (ENGAGE C-AXIS) (Engage C-Axis)
G00 G54 X3.0. C0. Y0. Z1.
G00 X1.5 Z0.5
M08
S500
G18 G95 C0 Z-0.5 R0.5 F0.05
G18 G95 C120. Z-0.5 R0.5 F0.05
G18 G95 C240. Z-0.5 R0.5 F0.05
G00 G80 Z0.5 M09
M135 (LIVE TOOL STOP)
G28 H0. (Unwind C-Axis)
M155 (C AXIS DISENGAGE)
G00 G54 X6. Y0 Z1.
G18 (Return to XZ plane)
G99 (Inches per minute)
M01
M30
%
```

Fine Spindle Control

Introduction

Many uses of live tooling involve holding the spindle still while performing a cut with the live tool. For certain types of operations, however, it is necessary to move this spindle in a controlled manner while cutting with the live tool. This section of the manual is a guide to the G-codes that are available to perform Fine spindle Control.

Uses for Fine Spindle Control

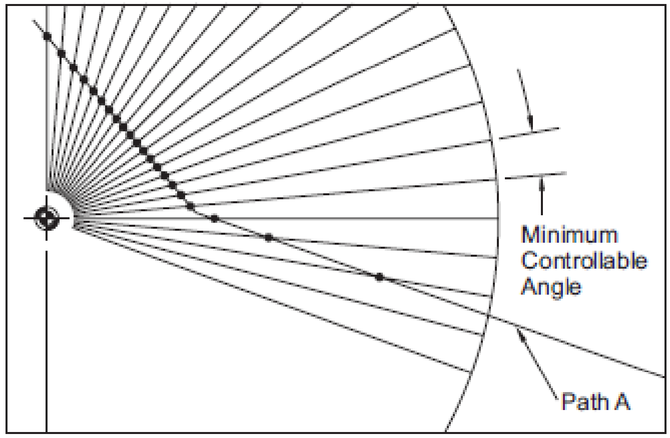
Fine Spindle Control (FSC) is most commonly used to create features on or near the face of a part, such as grooves, slots, and flat surfaces. Typically an end mill pointing along the Z axis is used to perform the cutting, after pilot holes are drilled. Live tooling is almost always required in order to use FSC. Single point turning is not recommended as the surface feet per minute required is too high for the FSC function.

Limitations of Fine Spindle Control

The primary function of the spindle is to turn rapidly. The introduction of G codes for FSC does not change the mechanical design of the spindle motor. Therefore, you should be aware of certain factors that apply when the spindle is turning at very low torque. This limits the depth of cut that can practically be performed be performed with the live tool while the spindle is not locked.

In many cases you will want to track the motion of the spindle with motion in the X axis. The spindle was designed to turn rapidly, rather than precisely. Because of this, the accuracy with which the position of the spindle is known is .045 degrees. This limit also applies to positioning the spindle in general. This also has an effect when trying to perform cuts that are close to centerline.

The number of control points depends on radius and direction of cutter path. Cutter paths with a large radius and a shallow angle towards the center will result in few control points. See Path A below.



G05 Fine Spindle Control motion Group 00

(NOTE: This G-code is optional and is used for live tooling on older machines. Machines 2014 or newer do not have this feature.)

R - Angular motion of the spindle, in degrees.
F - Feed Rate of the center of the tool, in inches per minute.
U - Optional X-axis incremental motion command.
W - Optional Z-axis incremental motion command.
X - Optional X-axis absolute motion command.
Z - Optional Z-axis absolute motion command.

This G code is used to specify a precise motion of the spindle, and is intended to be used for slotting. Any motion specified along the X and Z axes tracks the spindle motion. Currently, the resolution of the R code value is .045 degrees.

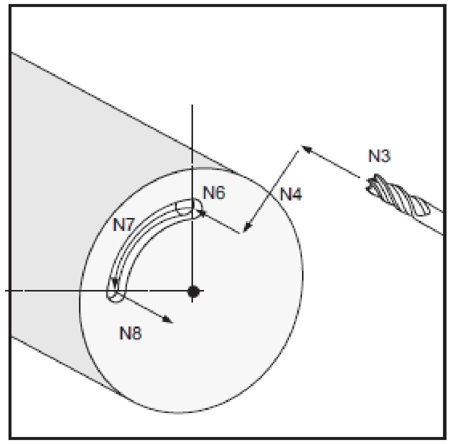
The rotational speed of the spindle will remain constant throughout each G5 cut. If there is motion along the X axis during the G05, the actual feed rate will vary. The spindle speed is determined by looking at the greatest X value encountered during the cut. Therefore the specified feed rate will not be exceeded at any point along the cut.

The largest feed per revolution value that can currently be specified is approximately 14.77. This means that G5 motions with small R motions relative to X or Z motions will not work. For example, an R motion of 1.5 degrees, the largest X or Z motion that can be specified is 14.77 * 1.5 / 360 = .0615 inches. Conversely, an X or Z motion of .5 inches must have an R travel of at least .5 * 360 / 14.77 = 12.195 degrees.

Example 1

Simple Face Slot with G05

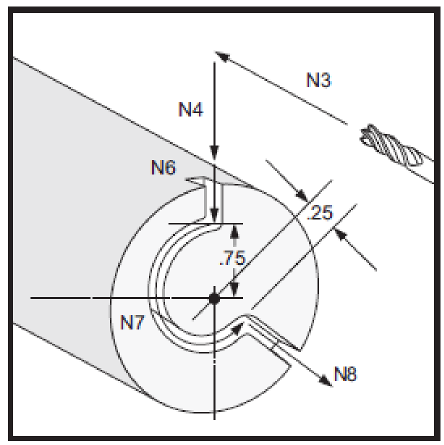
(Assume pilot hole is already drilled.)




```
N1   T303                       (Small End Mill)
N2   M19                        (Orient Spindle)
N3   G0 Z.5
N4   G0 X1.
N5   G133 P1500
N6   G98 G1 F10. Z-.25          (Plunge into pre-drilled hole)
N7   G5 R90.  F40.              (Make slot)
N8   G1 F10. Z.5                (Retract)
N9   G135
N10  G99 G28 U0 W0
```

Example 2

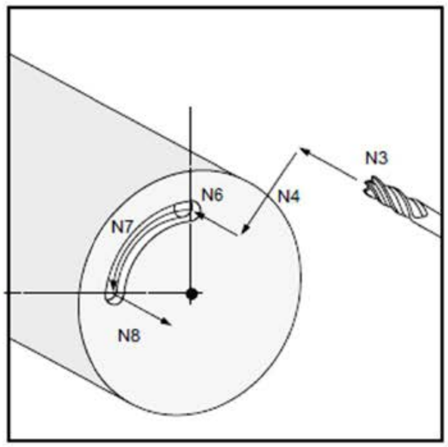
Simple Cam with G05




```
N1   T303                       (Small End Mill)
N2   M19
N3   G0 Z-.25
N4   G0 X2.5                    (Approach 2" dia. stock)
N5   M133 P1500
N6   G98 G1 X1.5 F40.           (Cut to top of cam)
N7   G5 R215. X.5 F40.          (Cut Cam)
N8   G1 X2.5 F40.               (Cut out of cam)
N9   M135
N10  G99 G28 U0 W0
```

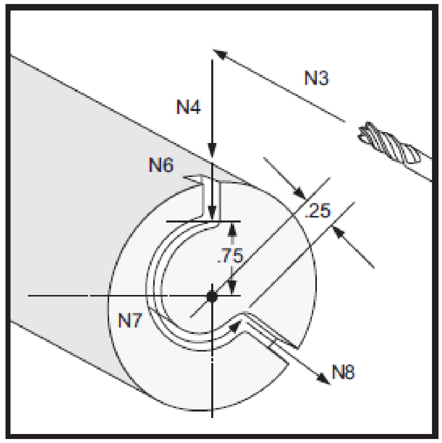
Axial or Face Milling

Slots may be created using the X and C axis. Illustration below shows creating an 1" diameter face ¼" wide slot just using the C axis in G18. The feed rate of the C-axis is in in/min. The Haas C-axis calculates the spindle rotation speed for a given feed rate by using the value in **setting 102 C-axis diamete**r. For the below program setting #102 should be set to 1 which is the diameter the tool is cutting at.



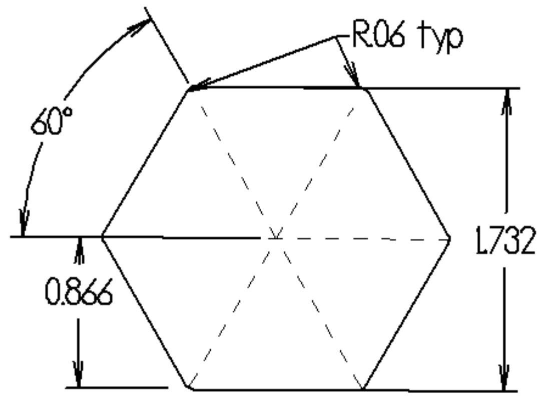

```
%
O00054
T707 (1/4 " END MILL)
G00 G54 G18
M133 P2000 (LIVE TOOLING FORWARD)
M154 (ENGAGE C-AXUS)
C0
N3 G00 G98 (feed/min) X2.0 Z.1
N4 X1.0
N6 G01 Z-0.1 F6.
N7 C90. F40.
G00 Z0.5
M155 (DISENGAGE C AXIS)
M135 (LIVE TOOLING OFF)
G53 X0
G53 Z0
M30
%
```

Below a simple cam is cut with X and C axis movement. Here setting #102 is at 1.5 the largest diameter the cam is cut at.




```
%
O00055 (SIMPLE CAM WITH X AND C)
T707
G54
M133 P2000
M154 (Engage C-axis)
N3 G00 G98 Z-0.25 (feed/min)
C0
N4 G00 X2.5 (2.IN DIAM STOCK)
N6 G01 X1.5 F40.
N7 C215. X0.5 F40.
N8 G01 X2.5 F40.
G00 Z0.5
M155
M135
G53 X0
G53 Z0
M30
%
```

When mill programming on the face of the part all X values must be in diameters. It is best to approach the part from the X direction and remember that the limit on the Y axis is +/- 2.0 radially. Note the C axis is not used and M10 clamps or breaks the spindle. All movement is in the X, Y and Z axis. **The following part however cannot be made**. This is because the **center of the live axial VDI tooling can only go .37 inches** on DSL 30Y and .401 inches on a STL20Y past centerline. Diametrically this is about .75 inch. This limits the use of milling using just the X, Y and Z axis.



%
O00017 (G17 AXIAL MILL WITH Y)
(CUTS 1.732 Hex)
(WITH .06 IN CORNER RAD)
(2.0 ROUND STOCK)
(T1 = .5IN ENDMILL)
(SET TOOL TO .25 RADIUS)
(ON TOOL OFFSET PAGE)
G53 G00 Y0.
G53 G00 X0.
G00 G54
M10 (CLAMP MAIN SPINDLE)
G17 (SELECTS G17 XY PLANE)
T101
G97 M133 P3000
G98 (IN PER MIN)
N1 X2.70 Y0
N2 Z0.1 (CLEARANCE PLANE)
N3 G01 Z-0.25 F10. (Z FINAL DEPTH)
G01 G41 X2.0 Y0
X1.0346 Y-.8360
G2 X.9306 Y-.866 R.06
G1 **X-.9306** Y-.866 **(Over Travel alarm in X)**

```
G2 X-1.0346 Y-.8360 R.06
G1 X-1.9654 Y-.03
G2 X-1.9654 Y.03 R.06
G1 X-1.0346 Y.8360
G2 X-.9308 Y.866 R.06
G1 X.9308 Y.866
G2 X1.0346 Y.8360 R.06
G1 X1.9654 Y.03
G2 X1.9654 Y-.03 R.06
G1 G40 X1.35 Y0
M135
G53 X0
G53 Y0
M30
%
```

G112 Cartesian to Polar Programming

G112 XY to XC interpolation (Group 04)

Polar coordinates are specific to rotary applications. Position is based on C angular degrees from a reference line. Radius is defined by X.

The G112 Cartesian to Polar coordinate transformation feature allows the user to program subsequent blocks in Cartesian XY coordinates, which the control automatically converts to polar XC coordinates.

While it is active, **G17** XY plane is used for G01 linear XY strokes and G02 and G03 for circular motion. X, Y position commands are converted into rotary C-axis and linear X-axis moves. See figure below. Note the centerline of the part is X0,Y0 in Cartesian Coordinate. See figure below.

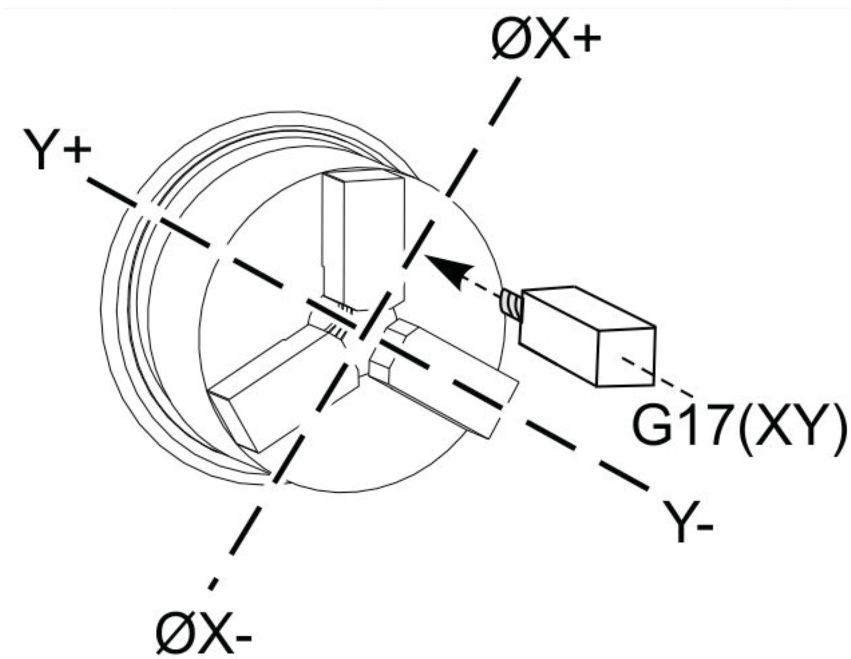
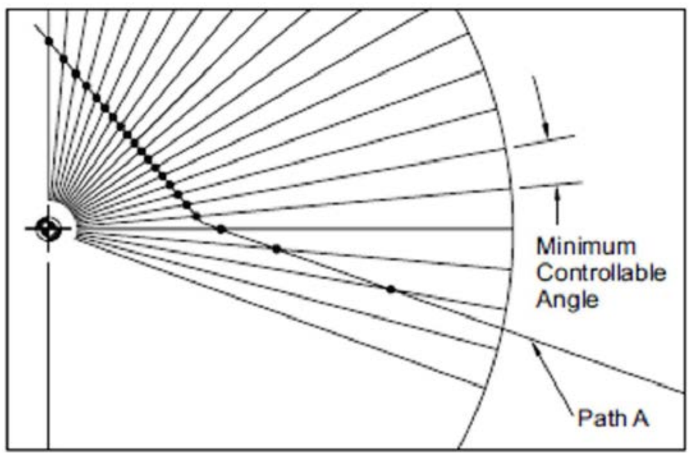


Illustration above taken from" Y-AXIS LATHE APPLICATIONS TRAINING", AP-100 Rev A Dec 2012 Haas Automation.

Note that mill-style Cutter Compensation becomes active when G41 is used. Cutter Compensation (G41, G42) must be canceled (G40) before exiting G112.

Note that no Y axis movement is made when using G112. All the motions of the machine are in X and C axii. When using G41 the radius of the tool that is used is put in the radius column in tool geometry. Also all negative X coordinates are converted to positive X coordinates. Then one doesn't have to worry about going below the centerline of the part.

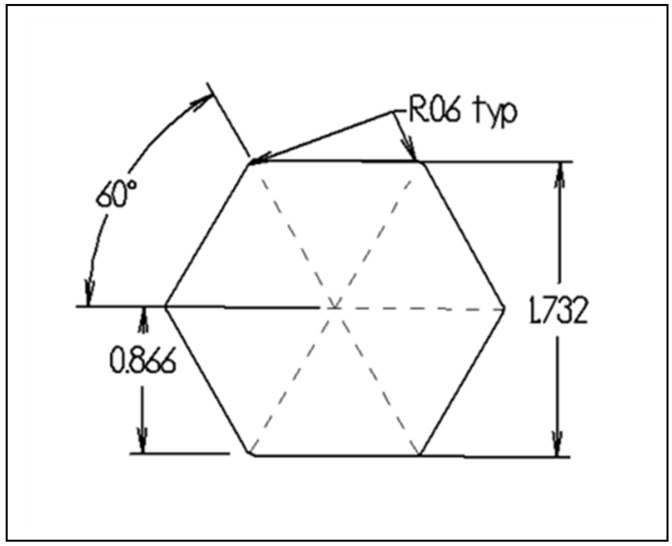
Feed rates will remain constant when using **G112**. What ever feed rate is called out in In/min the motion of the C-axis will be calculated so the feed rate called will be constant. As can be seen from the following figure at a constant feed rate in in/min the C-axis must turn faster when the tool is closer to centerline of the part than when it is farther away.



Normally a straight line would require many points to define the path as above in polar coordinates and each point requiring a different feed rate. However, in Cartesian coordinates , only end points are necessary. This feature allows face machining programming in the Cartesian coordinate system with simple in/min feed rates, **G98**.

G112 C-axis programming converts X,Y commands into rotary C-axis and linear X-axis moves. Another term for C-axis programming is Cartesian to Polar coordinate programming. Cartesian to Polar coordinate programming greatly reduces the amount of code required to command complex moves. On the next page the 1 .732 Hex may be cut using the following program with G112. Here the tool never crosses the centerline of the part in X.

Axially Cutting 1.732 Hex using G112

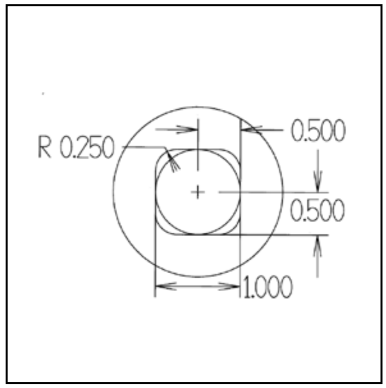



```
%
O00018 (G112)
(CUTS 1.732 Hex)
(WITH .06 IN CORNER RAD)
(2.0 ROUND STOCK)
(T1 = .5IN ENDMILL)
(SET TOOL TO .25 RADIUS)
(ON TOOL OFFSET PAGE)
G53 G00 Y0.
G53 G00 X0.
G00 G54
T101
M154 (ENGAGE C AXIS)
G28 H0 (HOME C AXIS)
G97 M133 P3000
G98 (IN PER MIN)
G17 (SELECTS G17 XY PLANE)
G112 (XY-XC INTERPOLATION)
N2 Z0.1 (CLEARANCE PLANE)
N3 G01 Z-0.25 F10. (Z FINAL DEPTH)
G00 X1.5 Y0
G01 G41 X1. Y0
X0.9827 Y-0.03
G01 X0.5173 Y-0.836
G02 X0.4653 Y-0.866 R0.06
G01 X-0.4654 Y-0.866
G02 X-0.5173 Y-0.836 R0.06
G01 X-0.9827 Y-0.03
G02 X-0.9827 Y0.03 R0.06
G01 X-0.5173 Y0.836
G02 X-0.4654 Y0.866 R0.06
G01 X0.4654 Y0.866
G02 X0.5173 Y0.836 R0.6
G01 X0.9827 Y0.03
G02 X0.9827 Y-0.03
G01 G40 X1.35 Y0
G113 (CANCEL G112)
G18 (X-Z PLANE)
G0 G28 H0
G99 M135
M155 (DISENGAGE C-AXIS)
G00 G53 X0
G00 G53 Y0
M30
%
```

Programming Notes

Programmed moves should always position the tool centerline with reference to the center line of the part. Tool paths should never cross the spindle centerline. If necessary re-orient the program so the cut does not go over the center of the part. Cuts that must cross spindle center can be accomplished with two parallel passes on either side of spindle center.

This program creates a 1” square with .25 radiuses .075 deep from face using a 3/8” end mill.




```
O151 (MILL 1" SQUARE)
N1 M01 (3/8 END MILL)
G00 G40 G99
G53 G00 Z-5.
G53 G00 X0.
N3 M01
T101 ( .375 DIA. E.M. )
M154 ( ENGAGE C-AXIS )
G28 H0
M133 P2650
G59 G00 Z0.15 C0.
G59 G00 X0 Y0.
G98
G17 ( X - Y PLANE )
G112 ( X-Y TO X-C INTERPOLATION )
G01 Z-0.075 F20.
G41 X.5 F11.25
Y.25
G03 X.25 Y.5 R.25
G01 X-.25
G03 X-.5 Y.25 R.25
G01 Y-.25
G03 X-.25 Y-.5 R.25
G01 X.25
G03 X.5 Y-.25
G01 Y0
G40 X0
G00 Z0.25 M09
```

```
G113 ( CANCEL G112 )
G18 ( X - Z PLANE )
G99 M135
G53 G00 X0.
G53 G00 Z-5.
M30
```

Note in the program that the X and Y values are **radial values**. **G98** in/min is called out and the **G17** (X,Y plane selection) must be called out. Also the tool diameter is compensated using a **G41**.

By contrast in **Fanuc code** instead of a **G112** a G12.1 is used. Also in Fanuc any G17,G18 and G19 codes are cancelled automatically. In Fanuc the Y axis is programmed as a C value. C becomes a virtual Y. The X axis programmed as an X value. These values are **diameters** in **Fanuc** compared to **radius** in **Haas**. Also in Fanuc a G1 code must be turned on before a G12.1 is called out. No G0's are allowed after the G12.1 is called out in Fanuc. G13.1 cancels G12.1 in Fanuc. Feed rates in Fanuc G12.1are similar to feed rates in Haas code G112, in/minute.

Radial or Cross Milling and Drilling

G75 O.D./I.D. Grooving Cycle (Group 00)

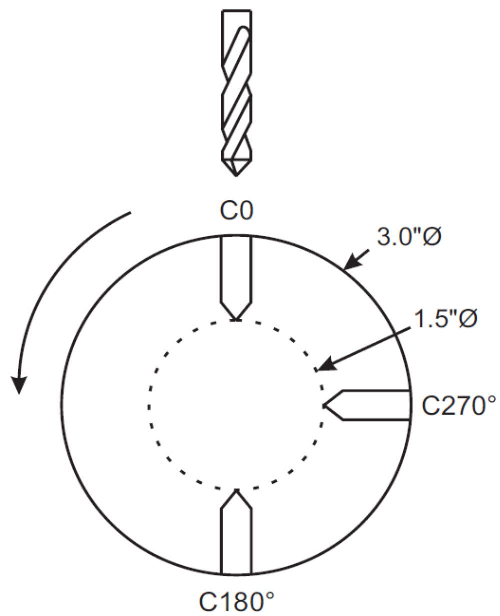
Can be used to drill holes on the outside diameter of parts. Machine must be put in **G19** plane to work.

G19 G75 X1.5 I0.25 F6

*X	X-axis absolute location total pecking depth (diameter)
*I	X-axis size of increment between pecks in a cycle (radial measure)
*F	Feed rate in Inches per minute (G98) active

Program Example

```
(LIVE DRILL - RADIAL)
T101
G19
G98
M154    (Engage C-axis)
G00 G54 X6. C0. Y0. Z1.
G00 X3.25 Z0.25
G00 Z-0.75
G97 P1500 M133
M08
G00 X3.25 Z-0.75
G00 C0.
G19 G75 X1.5 I0.25 F6.
G00 C180.
G19 G75 X1.5 I0.25 F6.
G00 C270.
G19 G75 X1.5 I0.25 F6.
G00 G80 Z0.25 M09
M135
M155
M09
G00 G28 H0.
G00 X6. Y0. Z3.
G18
G99
M00
M30
%
```

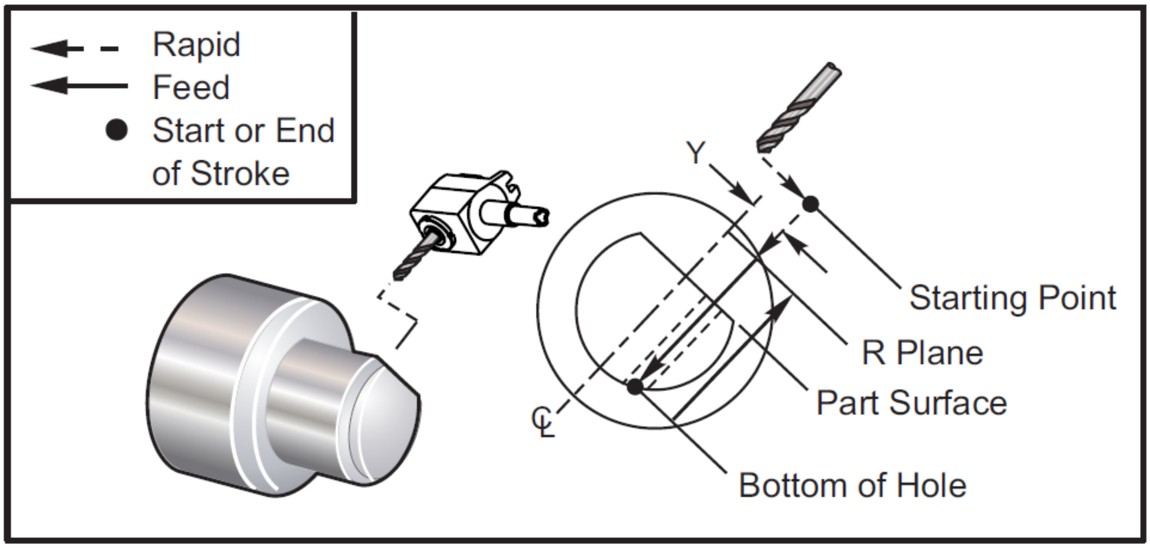


Radial Canned Cycles G241, 242 and 243 work similar to G81, G82 and G83. Machine must be in **G98 inch per minute mode.** Machine must be put in **G19** plane selection>

G241 Radial Drill Canned Cycle (Group 09)

G241 X2.1 Y0.125 Z-1.3 C35. R4. F20

C   C-axis absolute motion command
F   Feed Rate Inch/mine
R   Position of the R plane (diameter)
*X   Position of bottom of hole (diameter)
*Y   Y-axis absolute motion command
*Z   Z-axis absolute motion command
* indicates optional



G241 Radial Drill Canned Cycle

(G241 - RADIAL DRILLING) Example

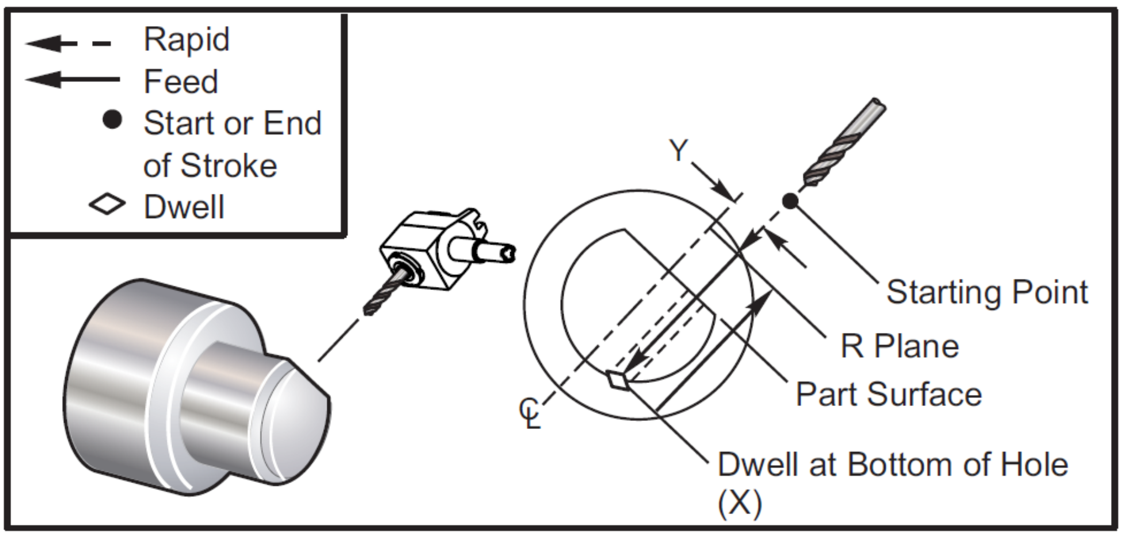
```
O1
G54  (Work Offset G54)
G00 G53 Y0  (Home Y-axis)
G00 G53 X0 Z-7.
T303
M154  (Engage C Axis)
M133 P2500  (Live Tooling On, 2500 RPM)
G19 (Y-Z Plane Selection)
G98 (IPM)
G00 X5. Z-0.75 Y0
G241 X2.1 Y0.125 Z-1.3 C35. R4. F20.  (Drill to X 2.1)
X1.85 Y-0.255 Z-0.865 C-75.
G00 G80 Z1.
M135 (Stop live tool spindle)
G00 G53 X0 Y0
```

G242 Radial Spot Drill Canned Cycle (Group 09)

Drill and dwell canned cycle

C   C-axis absolute motion command
F   Feed Rate
P   The dwell time at the bottom of the hole
R   Position of the R plane (Diameter)
*X   Position of bottom of hole (Diameter)
*Y   Y-axis motion command
*Z   Z-axis motion command
* indicates optional

This G code is modal. It remains active until it is canceled (G80) or another canned cycle is selected. Once activated, every motion of Y and/or Z will execute this canned cycle.



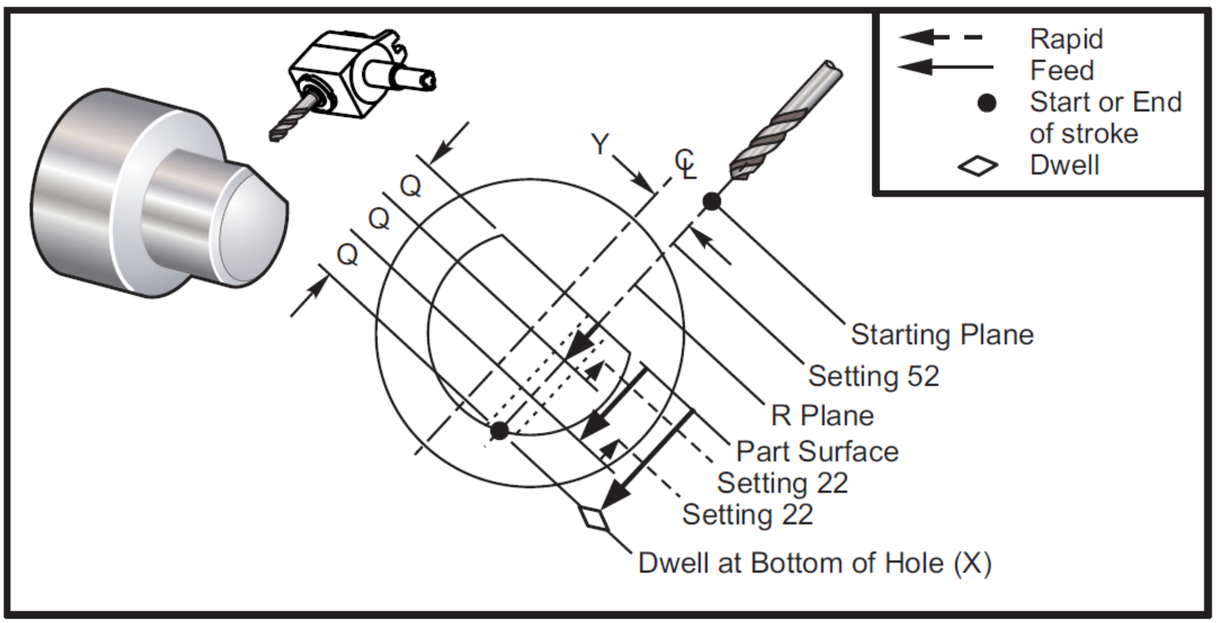
G242 Radial Spot Drill Canned Cycle

Program Example (drill and dwell .5 second)

```
G54 (Work offset G54)
G00 G53 Y0 Home Y-axis)
G00 G53 X0 Z-7.
T303
M154 (Engage C Axis)
M133 P2500 (2500 RPM)
G19 (Y-Z Plane Selection)
G98 (IPM)
G00  X5.  Z-0.75 Y0
G242  X2.1  Y0.125  Z-1.3 C35. R4. P0.5  F20. (Drill to X 2.1)
X1.85  Y-0.255 Z-0.865 C-75. P0.7
G00  G80  Z1.
M135 (Stop live tool spindle)
G00 G53 X0. Y0.
G00 G53 X0 Z-7.
M30
```

G243 Radial Normal Peck Drilling Canned Cycle (Group 09)

C C-axis absolute motion command
F Feed Rate (G98 In/mn)
*I Size of first cutting depth
*J Amount to reduce cutting depth each pass
*K Minimum depth of cut
*P The dwell time at the bottom of the hole
*Q The cut-in value, always incremental
R Position of the R plane (Diameter)
*X Position of bottom of hole (Diameter)
*Y Y-axis absolute motion command
*Z Z-axis absolute motion command
* indicates optional



G243 Radial Normal Peck Drilling Canned Cycle

Programming Notes

If I, J, and K are specified, a different operating mode is selected. The first pass will cut in the value of I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K. Do not use a **Q** value when programming with **I,J,K**.

Setting 52 changes the way G243 works when it returns to the R-plane. Usually the R plane is set well outside the cut to insure that the chip clearing motion allows the chips to clear the hole. However, this is wasted motion when first drilling through this “empty” space. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled. When the clear move to R occurs, the Z will be moved past R by this value in setting 52.

Setting 22 is the amount to feed in X to get back the same point at which the retraction occurred.

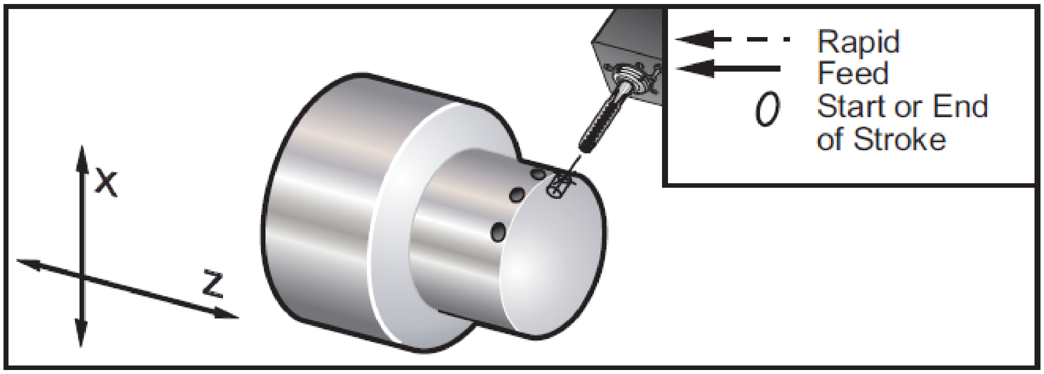
Program Example

(G243 - RADIAL PECK DRILLING USING Q)

```
G54  (Work offset G54)
G00 G53 Y0  (Home Y-axis)
G00 G53 X0  Z-7.
T303
M154 (Engage C Axis)
M133 P2500 (2500 RPM)
G19
G98 (IPM)
G00 X5. Z-0.75 Y0
G243  X2.1  Y0.125  Z-1.3  C35.  R4.  Q0.25  F20. (Drill to X 2.1)
X1.85  Y-0.255  Z-0.865  C-75.  Q0.25
G00 G80  Z1.
M135 (Stop live tool spindle)
G00  G53  X0.  Y0.
G00  G53  X0  Z-7.
M00
```

(G243 - RADIAL WITH I,J,K PECK DRILLING)

```
G54  (Work offset G54)
G00  G53  Y0 (Home Y-axis)
G00  G53  X0  Z-7
T303
M154  (Engage C Axis)
M133  P2500  (2500 RPM)
G19
G98  (IPM)
G00  X5.  Z-0.75  Y0
G243  X2.1  Y0.125  Z-1.3  I0.25  J0.05  K0.1  C35.  R4. F5. (Drill to X 2.1)
X1.85  Y-0.255  Z-0.865  I0.25  J0.05  K0.1  C-75.
G00  G80  Z1
M135
G00  G53  X0  Y0
G00  G53  Z-7
M00
```



G195 / G196 Live Tooling Rigid Tapping (Diameter)

G195 Live Tool Radial Tapping (Diameter) (Group 00)

G195 cycle operates differently than G241-243. It must be called up after each location change. Also feed rate **must be in G99** (in/rev) as opposed to G98 (in/min). **Only** a speed value is given as a **SXXXX** but the live tooling command is omitted (M133 PXXXX). Live tooling must be cancelled when the tapping is done with M135.

Also note some manufactures gearing is different on their radial live tooling. When using **Heimetec radial live tools the tool rotates in the opposite direction**. So instead of G195 for right hand threads a G196 must be used.

F **Feed Rate per revolution (G99)**
*U X-axis incremental distance
*X X-axis motion command
*Y Y-axis motion command
*Z Z position prior to drilling
*indicates optional

Program Example G195 using C-axis

```
(LIVE TAP - RADIAL)
T101
G19
G99
M154 (Engage C-Axis)
G00 G54 X6. C0. Y0. Z1.
G00 X3.25 Z0.25
G00 Z-0.75
G00 C0.
S500
G19 G195 X2. F0.05
G00 C180. (Index C-Axis
G19 G195 X2. F0.05
G00 C270. (Index C-Axis)
G19 G195 X2. F0.05
G00 G80 Z0.25 M09
M135
M155
M09
G00 G28 H0.
G00 X6. Y0. Z3.
G18
G99
M30
```

Below is a program example of G195 using M19

```
O00800
N1 T101 (RADIAL 1/4-20 TAP)
G99 (Necessary for this cycle)
G00 Z0.5
X2.5
Z-0.7
S500 (rpm should look like this, cw direction)**
M19PXX (Orient spindle at desired location)
M14(Lock spindle up)
G195 X1.7 F0.05 (thread down to X1.7)
G28 U0
G28 W0
M135 (Stop Live tooling spindle)
M15 (Unlock Spindle brake)
M30
%
```

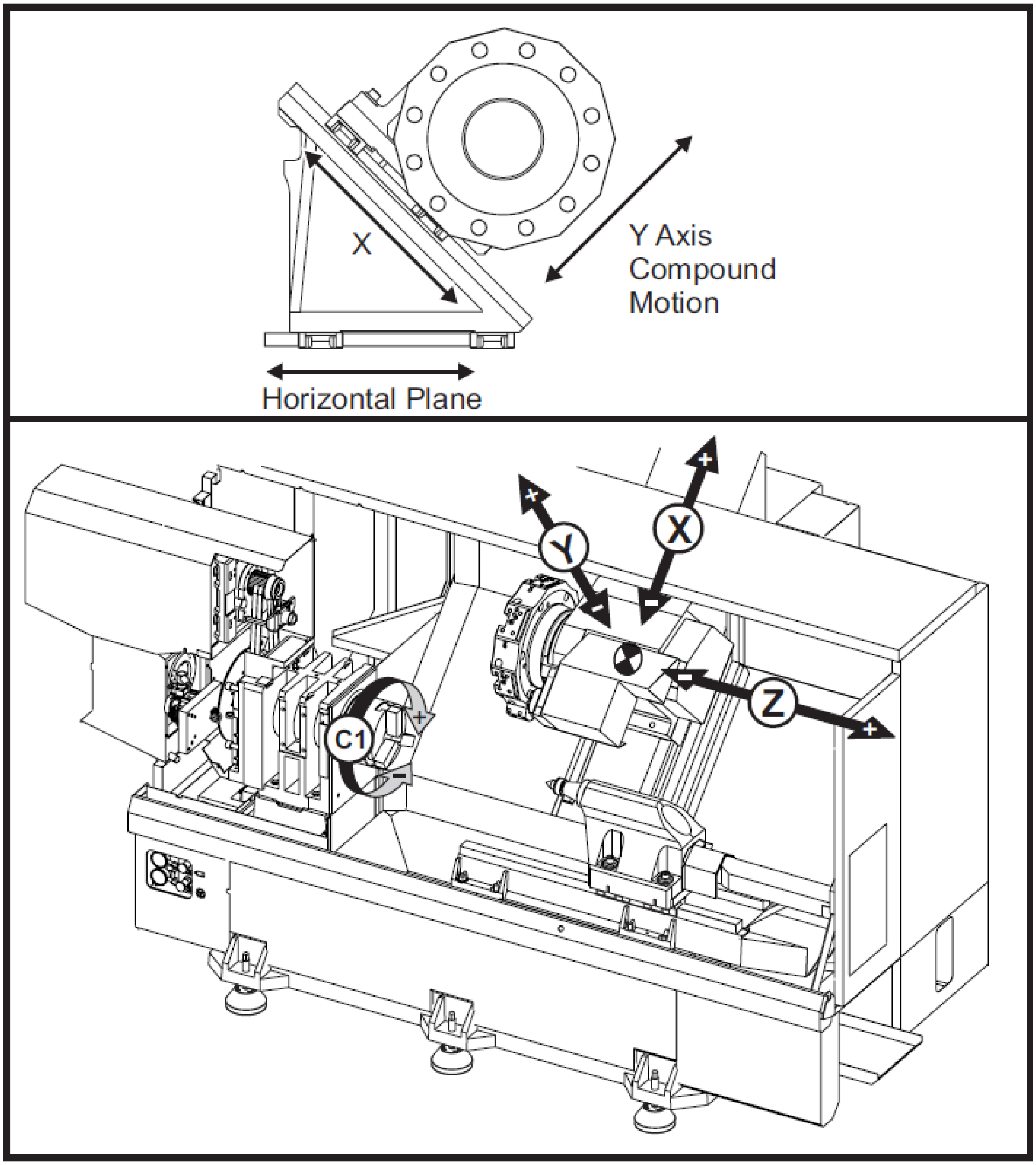
G196 Reverse Live Tool Radial Tapping (Diameter) (Group 00)

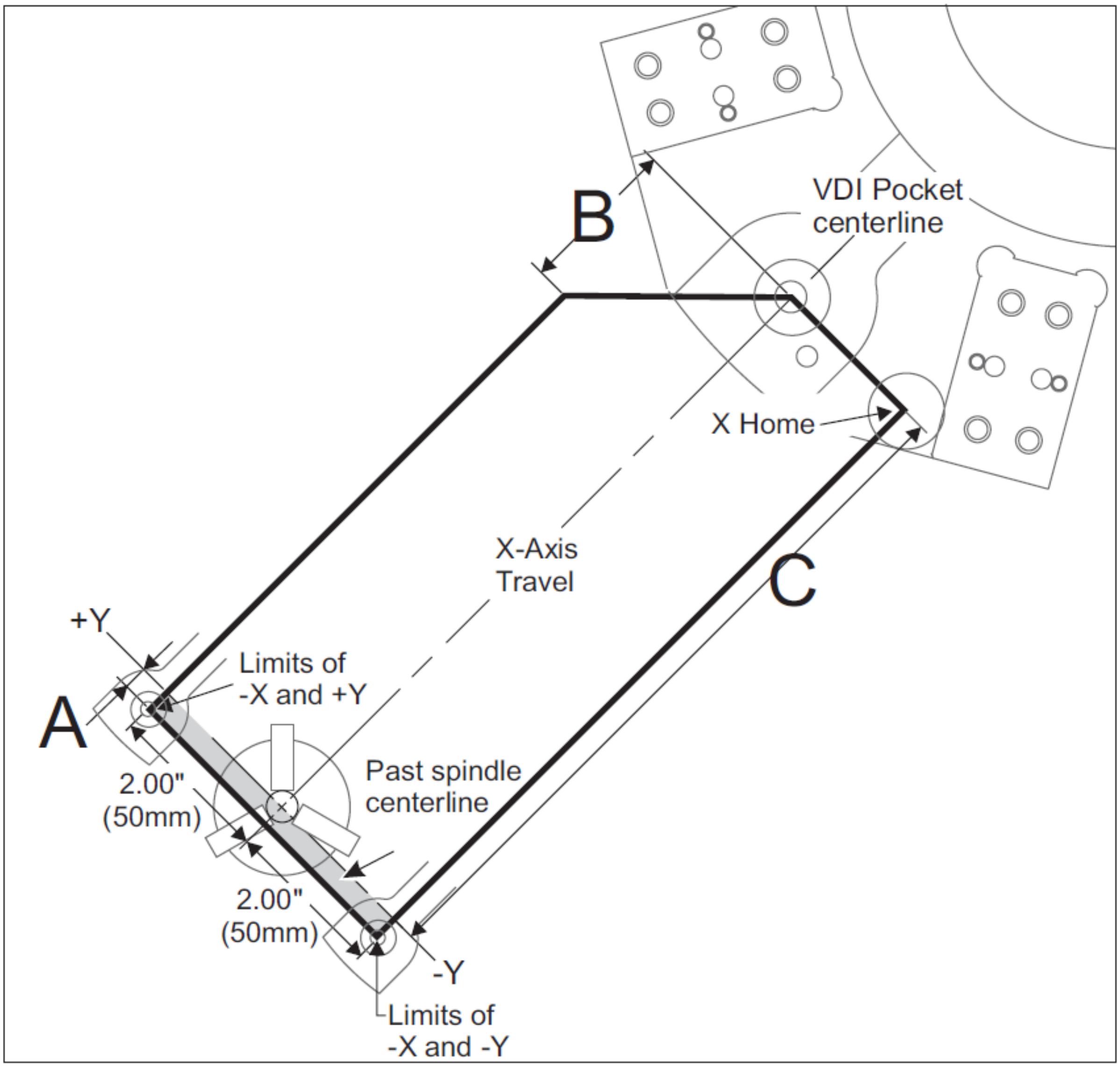
F Feed Rate per revolution (G99)
*U X-axis incremental distance
*X X-axis motion command
*Y Y-axis motion command
*Z Z position prior to drilling
*Indicated optional

These G codes perform live tooling radial or vector tapping on a lathe; they do not permit an "R" plane.

Y-Axis

The Y-axis moves tools perpendicular to the spindle center line. This motion is achieved by a compound motion of the X-axis and Y-axis ball screws. Also see G17 XY plane and G19 YZ plane for programming information.





Dimensions from VDI pocket centerline for Axial tools				
	ST-20Y with VB / VDI	ST-20SSY with 24-tool Hybrid	ST-30Y/DS-30Y with VB / VDI	ST-30SSY / DS-30SSY with 24-tool Hybrid
A	0.30" (8 mm)	1.61" (41 mm)	1.55" (39 mm)	2.86" (80 mm)
B	2.00" (50 mm)	2.00" (50 mm)	2.00" (50 mm)	2.00" (50 mm)
C	9.00" (228 mm)	7.69" (195 mm)	10.95" (278 mm)	9.64" (245 mm)
Maximum Cutting Diameter for Axial Tools				
	14.00 (356 mm)	11.38" (289 mm)	17.90" (455 mm)	15.28" (388 mm)

Above illustrate the Y-Axis Travel Envelope. Note as the machine nears X home the movement in Y becomes truncated. Otherwise the movement in Y is +/- 2.0 from centerline. The next few pages illustrate the travel envelopes of the Y-axis lathes.

Y-Axis Travel Envelopes

The opposite pages illustrate the travel envelopes of the Y-axis lathes. The Y-axis travel limits are shown on the following pages relative to the VDI tool pocket centerline and the spindle centerline. The size and position of the available work envelop changes with the length of radial live tools. When setting up tooling consider the following:

- Work piece diameter
- Tool extension (radial tools)
- Required Y-axis travel from the centerline

Y-Axis Lathe with VDI Turret

For standard axial tool holders, the centerline of the cutting tool will be available in the following work envelope illustration. The position of the work envelope will shift when using radial live tools. The length the cutting tool extends from the centerline of the tool pocket is the distance the envelope shifts. The opposite illustration demonstrates the work envelope in relation to the center of the VDI tool pocket.

Operation and Programming

The Y-axis is an additional axis on the lathes (if so equipped) that can be commanded and behaves in the same manner as the standard X and Z axis. There is no activation command necessary for Y-axis. It is available at all times when machine is in run or set-up mode.

The lathe will automatically return the Y-axis to spindle centerline after a tool change. Make sure the turret is correctly positioned before commanding rotation.

Standard Haas G and M codes are available when programming with Y-axis. Please refer to G and M code section of this manual for more information. Plane selection commands are necessary for Y-axis live tooling operations. This applies to both axial live tools (tool centerline parallel to the Z-axis) and radial live tools (tool centerline parallel to the X-axis). Please refer to G17, G18 and G19 code explanations in your machine manual. Mill type cutter compensation can be applied in both G17 and G19 planes when performing live tool operations. Cutter compensation rules must be followed to avoid unpredictable motion when applying and canceling the compensation. The Radius value of the Tool being used must be entered in the Radius column of the tool geometry page for that tool. The tool tip is assumed as “0” and no value should be entered.

Y-Axis Programming Recommendations & Examples

1) Command Axis home or to a safe tool change location in rapids using **G53**. Both axis can be commanded at the same time regardless of the positions of Y-axis and X-axis in relation to each other. All axes will move at the MAX possible speed toward commanded position and will not finish at the same time. If commanding the Y and X axes home using G28 the following conditions must be met and the described behavior expected.

- If X-axis is commanded home while the Y-axis is above spindle centerline (positive Y-axis coordinates), alarm 317 (Y over travel range) will be generated. Command Y-axis home first, then X-axis.

- If X-axis is commanded home and the Y-axis is below spindle centerline (negative Y axis coordinates), the X-axis will home and Y will not move.

- If both X-axis and Y-axis are commanded home using G28 X0 Y0 and the Y-axis is below spindle centerline (negative Y axis coordinates), the Y-axis will home first and the X-axis will follow.

2) Clamp the main and/or secondary spindles (if so equipped) anytime live tooling operations are being performed and C-axis is not being interpolated. Note that the brake will unclamp automatically anytime C-axis motion for positioning is commanded. Refer to C-axis, Live Tooling and M-code section for more information.

3) The following canned cycles can be used with Y-axis. Refer to the G-code section of this manual for more information.

   G18 Plane (Axial) Only Cycles:
   Drilling: G81, G82, G83, G85, G89
   Tapping: G95, G186

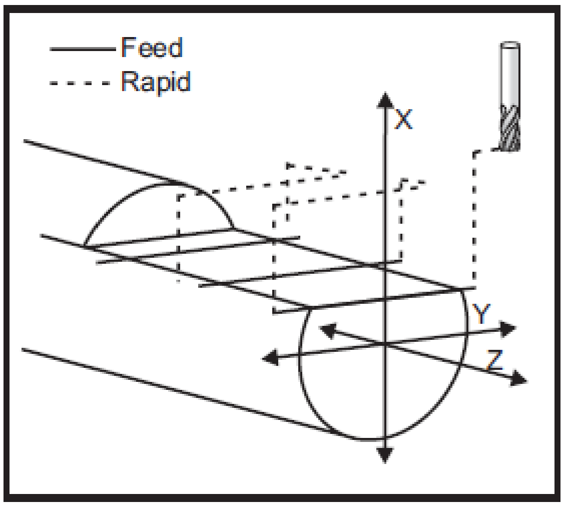
   G19 Plane (Radial) Only Cycles:
   Drilling: G75 (a grooving cycle), G241, G242, G243,
   Boring: G245, G246, G247, G248
   Tapping: G195, G196

The programs on pages 30-36 show examples of cross or radial drilling using y-axis capabilities of the machine.

Milling Flats with Y Axis

Program Example

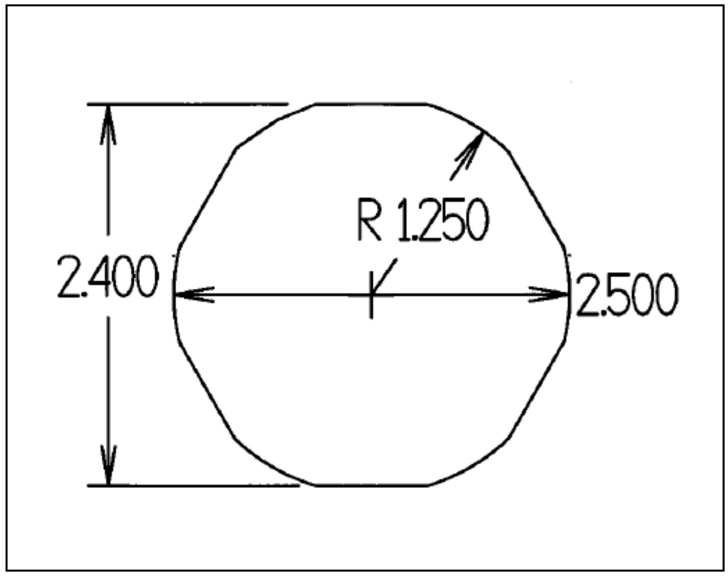
```
%
O02003
N20
(MILL FLAT ON DIAMETER 3.00 DIAMETER .375 DEEP)
T101 (.750 4 FLUTE ENDMILL)
G19 (SELECT PLANE)
G98 (IPM)
M154 (ENGAGE C-AXIS)
G00 G54 X6. C0. Y0. Z1. (RAPID TO A POSITION)
G00 C90. (ROTATE C AXIS TO 90 DEGREES)
M14 (BRAKE ON)
G97 P3000 M133
G00 X3.25 Y-1.75 Z0. (RAPID POSITION)
G00 X2.25 Y-1.75
M08
G01 Y1.75 F22.
G00 X3.25
G00 Y-1.75 Z-0.375
G00 X2.25
G01 Y1.75 F22.
G00 X3.25
G00 Y-1.75 Z-0.75
G00 X2.25
G01 Y1.75 F22.
G00 X3.25
G00 X3.25 Y0. Z1.
M15 (BRAKE OFF)
M135 (LIVE TOOL OFF)
M155 (DISENGAGE C-AXIS)
M09
G00 G28 H0.
G00 X6. Y0. Z3.
G18 (RETURN TO NORMAL PLANE)
G99 (IPR)
M01
M30
%
```



Example of using Y-axis to Mill 2.4 Hex on 2.5" Diameter Round Stock

```
O00101 ( BASIC MILLING SAMPLE )
( MACHINE WITH G58 )
( MILL SINGLE END )
G00 G40 G99
G53 G00 X0.
G53 G00 Z-5.
N9 M01 ( 1.0 DIA. 3FL. E.M. ) ( ER-32 RADIAL HOLDER )
```

Example of using Y-axis to Mill 2.4 Hex on 2.5" Diameter Round Stock *(Continued)*



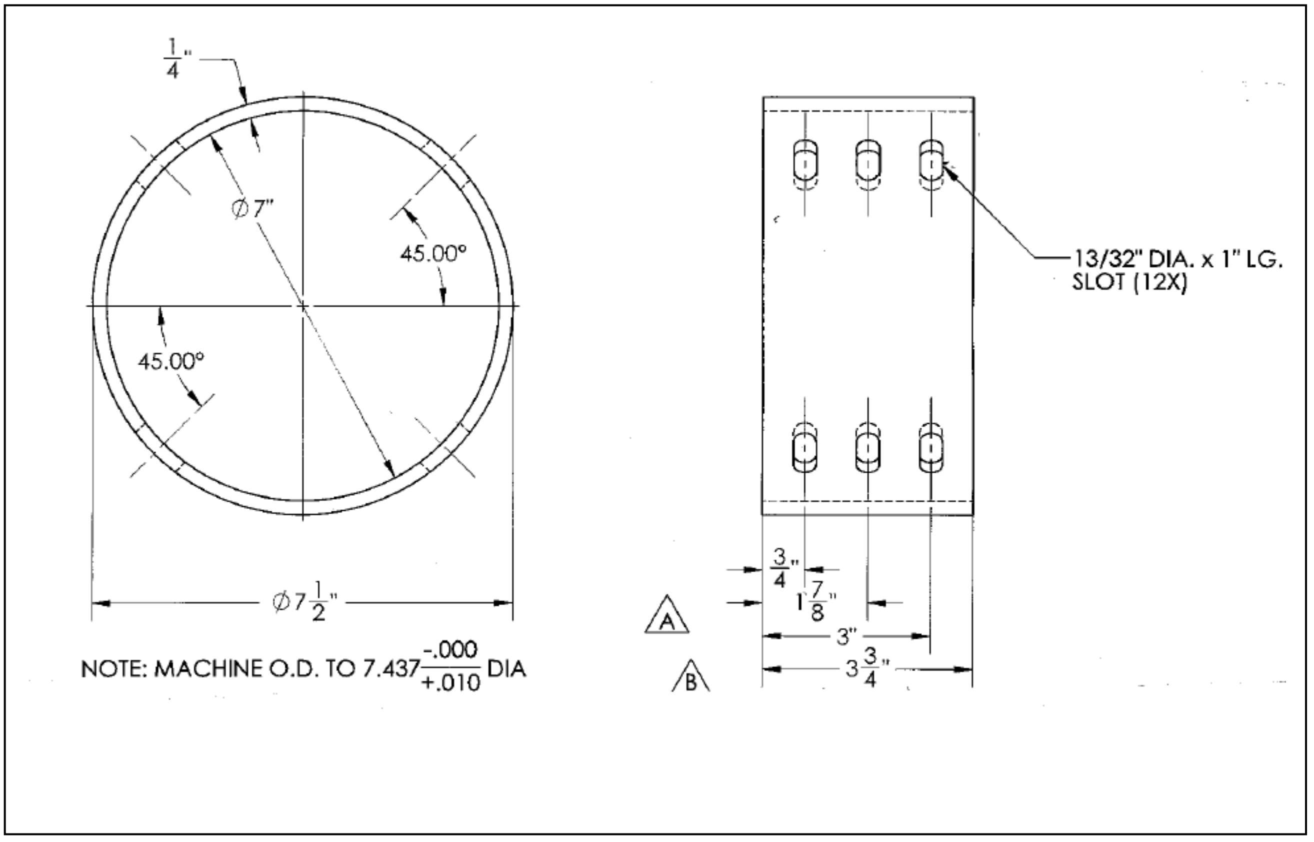

```
G53 G00 X0.
G53 G00 Z-5.
T909 ( 1.0 DIA. E.M. )
M154 ( ENGAGE C-AXIS )
G28 H0.
M133 P2650
G98
G58 G00 Z0. C0.
G58 G00 X3.5
G58 G00 Y-1.25
( C0.0 )
G01 X2.4 F20.
G01 Y1.25 F10.
G00 X3.5
( C60.0 )
G00 Y-1.25 C60.
G01 X2.4 F20.
G01 Y1.25 F10.
G00 X3.5
( C120.0 )
G00 Y-1.25 C120.
G01 X2.4 F20.
G01 Y1.25 F10.
G00 X3.5
( C180.0 )
G00 Y-1.25 C180.
G01 X2.4 F20.
G01 Y1.25 F10.
G00 X3.5
( C240.0 )
G00 Y-1.25 C240.
G01 X2.4 F20.
G01 Y1.25 F10.
G00 X3.5
( C300.0 )
G00 Y-1.25 C300.
G01 X2.4 F20.
G01 Y1.25 F10.
G00 X3.5 Z2. M09
G28 W0. H0.
G99 M135
G53 G00 X0.
G53 G00 Z-5.
M30
```

C-Axis Feed Rate

When C-Axis is engaged the **units for feed are inches per min**. On Haas Lathes Feed rates when the C-axis is engaged with M154 are determined by the diameter entered in **setting 102 (C-AXIS DIAMETER).** From the factory setting 102 is set at 1.00.

If one wants the units to be in degrees/minute the Haas lathe must be turned to metric and setting 102 set to 114.5. This value is same as $\frac{360}{\pi}$.

To calculate a given feed rate in inches per minute to degrees per minute refer to page 16. Noting figure on page 21 many feed rates need to be determined to keep a constant chip load on the tool.



EXERCISE #1: Calculate the feed rate in degrees/minute for the above slots. You are using a 13/32 Carbide, 4-flute end, mill cutting at a speed of 120 ft/min. with a chip load of .002 in/ min-tooth. One end of the slots has been predrilled to a size of ¼ in. Use the 7 ½" diameter to figure your feed rate.

The ¼ Drill feed rate is .003"/rev at surface feed of 120 Ft/min.

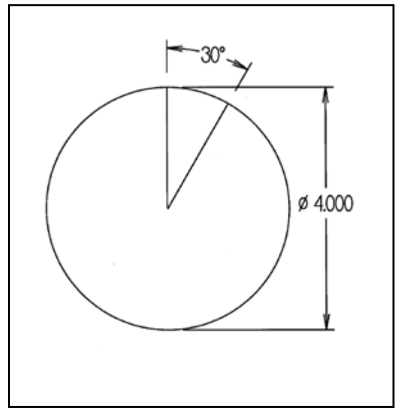
EXERCISE #2:

G241 AND C-AXIS MILLING EXERCISE

```
%O01111 (G124 DRILL RAD. SLOT MILLING)
G53 X0
G53 Z-5.0
T0303 (1/4" DRILL)
M___ (ENGAGE C AXIS)
M____ P1833 (TURN ON LIVE TOOLING)
G___ Y-Z PLANE SELECTION
G___ (IN/MIN)
G00 Z-.75
X7.7 Y0
G241 X___ Y0 Z-___ R___ C0 F____ (DRILL TO X6.9)
Z-1.875
Z-3.0
C____
Z-1.875
Z-.75
C___.
Z-1.875
Z-3.0
C180
Z-1.875
Z-.75
C270.
Z-1.875
Z-3.0
G00 G80 Z1.0 C0
M____ (LIVE TOOL STOP)
G53 X0
G53 Z-5.0

N5 (13/32 END MILL)
G00 G53 X0
G53 Z-5.0
T0303 (13/32 END MILL)
M154 (ENGAGE C AXIS)
M133 P1833 (TURN ON LIVE TOOLING)
G98 (IN/MIN)
C0
G0 X7.7 Z.3
Z-.75
G1 X6.9 F7.3
C____ F______
G0 7.7
```

```
C90.0
G1 X6.9 F7.3
C_____ F______
G0 7.7
C180.
G1 X6.9 F7.3
C189.072 F______
G0 7.7
C270.
G1 X6.9 F7.3
C279.072 F______
G0 7.7
G0 Z.4
M_____ (DISENGAGE C AXIS)
G18 ( X - Z PLANE )
G99 M_____ (LIVE TOOL STOP)
G53 G00 X0.
G53 G00 Z-5.
M30
%
```

Example of Calculation of Feed Rate in Degrees/Min for the C-Axis



1st Calculate Radial move in inches

Circumference of circle = Diameter x π
= 4 x π
= 12.566

Distance travelled over 30°=30/360 x 12.566
=1.047

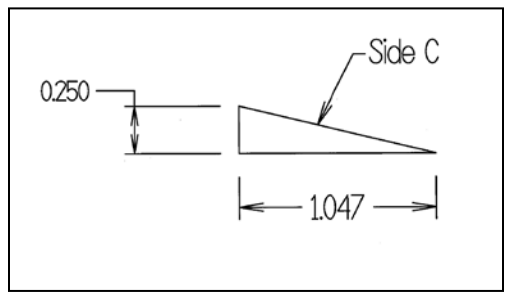
2nd Movement in Z if any, example .250

3rd Calculate total movement (side C)

$C= \sqrt{A^2 + B^2}$

$C=\sqrt{.25^2+1.047^2}$

C= 1.0762



4th Calculate RPM and Feed Rate in Inch/Minute

RPM = 3.82x(S ft/min)/ Diameter

IPM = FPT x T x RPM

From above equations as an example use 12.0 Inch/min

5th Calculate time to complete movement in C

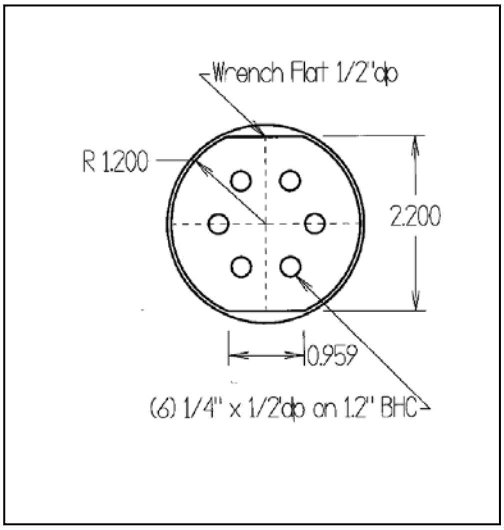
Minutes to C movement = C/ IPM
= 1.076/12.0
= .08971 minutes

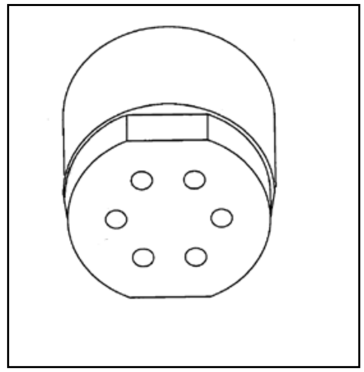
Feed Rate in Degrees/Min = Degrees traveled / time in minutes

= 30 degrees/ .08971 minutes
= 334.4 Degrees/Minute

EXERCISE #3:

Y-Axis and Face Drilling





Use ¾” four flute end mill in radial or cross live tool holder to cut ½’ flat. Run end mill at surface footage of 120 ft/min with chip load of .002”/rev-tooth.

Drill ¼” holes using a high speed drill with a face cutting or axial live tool. Use a G81 canned cycle. Run ¼” drill at 120 ft/min, .003”/rev.

```
O00102 ( Y AXIS AND FACE DRILLING EX )
( MACHINE WITH G58 )
( MILL SINGLE END )
G00 G40 G99
G53 G00 X0.
G53 G00 Z-5.
N9 M01 ( .75 DIA. 4FL. E.M. ) ( ER-32 RADIAL HOLDER )
G53 G00 X0.
G53 G00 Z-5.
T909 ( .     DIA        . )
M___ ( ENGAGE C-AXIS )
G28 H0
G00 C__.
M___ P___  (LIVE TOOLING FORWARD)
(G98
G58 G00 Z_____ C0.
G58 G00 X____
G58 G00 Y-_____
( C0.0 )
G01 X_____ F_____.
```

```
G01 Z_____
_____ (BRAKE ON)
G01 Y1.00 F4.9.
G00 X2.6
( C180.0 )
G00 Y-1.0 C____.
G01 X____ F4.9.
G01 Y_____0 F4.9.
_____ (BRAKE ON)
G00 X3.5 Z2. M09
G28 Y0. C0.
G99 M135
G53 G00 X0.
G53 G00 Z-5.
M01

T101 (1/4" DRILL)
G54
G00 X1.2 Z0.1
G18 (X-Z PLANE)
G98
M_____ (C-Axis engage)
C_____.
M_____P_______ (Live Tooling Drive Forward )
G___ Z____ F____
C_____
C_____
C_____
C_____.
C_____
G00 G80 Z0.1
M1____ (C-Axis Disengage)
M_____ (LIVE TOOL DRIVE STOP)
G53 X0
G53 Z0
M30
%
```

Solutions to Exercises

ercise P

1st calculate the radial distance of cut for the 13/32 End Mill
From print a 1" slot needs to be cut using 13/32 end mill.
The radial distance the end mill needs to go is (1 – 13/32) = **_____**"
The circumference of a circle is diameter x π
Circumference of tube = 7.5" x π = 23.562
Degrees travelled per *.5937"* cut = fraction of circumference x 360 degrees/circumference
= (.5937/23.562) x 360 degrees
= **de rees**

2nd calculate feed rate
RPM = 3.82 x Speed/ Diameter = 3.82 x 120 ft/min / .4062 = 1128 rev/min

Feed/Min = Rev/min x In/rev x # Teeth
= 1128 x .002"/min x 4
= **in in**

3rd Calculate time to complete cut
Time = distance/ Feed/min
= .5937/9.024
= **in**

Feed rate in De rees in = Degrees travelled/ time in minutes
= 9.072 degrees / .06579 minutes
= **de rees inute**

ercise P

```
%
O01111 (G241 DRILL RAD. SLOT MILLING)
G53 X0
G53 Z-5.
T1111 (1/4" DRILL)
M154 (ENGAGE C AXIS)
M133 P1833 (TURN ON LiVE TOOLING)
G98 (IN/MIN)
G19 (Y-Z PLANE SELECTION)
G00 G56 Z-0.75
X7.7 Y0
G241 X6.9 Y0 Z-0.75 R7.7 C0 F7.3 (DRILL TO X6.9)
Z-1.875
Z-3.
C90.
Z-1.875
Z-0.75
```

```
C180.
Z-1.875
Z-3.
C180
Z-1.875
Z-0.75
C270.
Z-1.875
Z-3.
G00 G80 Z1. C0
M135 (LIVE TOOL STOP)
G53 X0
G53 Z-5.

N5 (13/32 END MILL)
G00 G53 X0
G53 Z-5.
T1111 (13/32 END MILL)
M154 (ENGAGE C AXIS)
M133 P1833 (TURN ON LIVE TOOLING)
G98 (IN/MIN)
C0
G00 G56 X7.7 Z0.3
Z-0.75
G01 X6.9 F7.3 (F138.)
G01 C9.071 F13.
G00 X7.7

C90.
G01 X6.9 F7.3
C99.071 F13.
G00 X7.7

C180.
G01 X6.9 F7.3
C189.071 F13.
G00 X7.7

C270.
G01 X6.9 F7.3
C279.071 F13.
G00 X7.7
G00 Z0.4
M155 (DISENGAGE C AXIS)
G18 ( X - Z PLANE )
G99 M135 (LIVE TOOL STOP)
G53 G00 X0.
G53 G00 Z-5.
M30
%
```

ercise

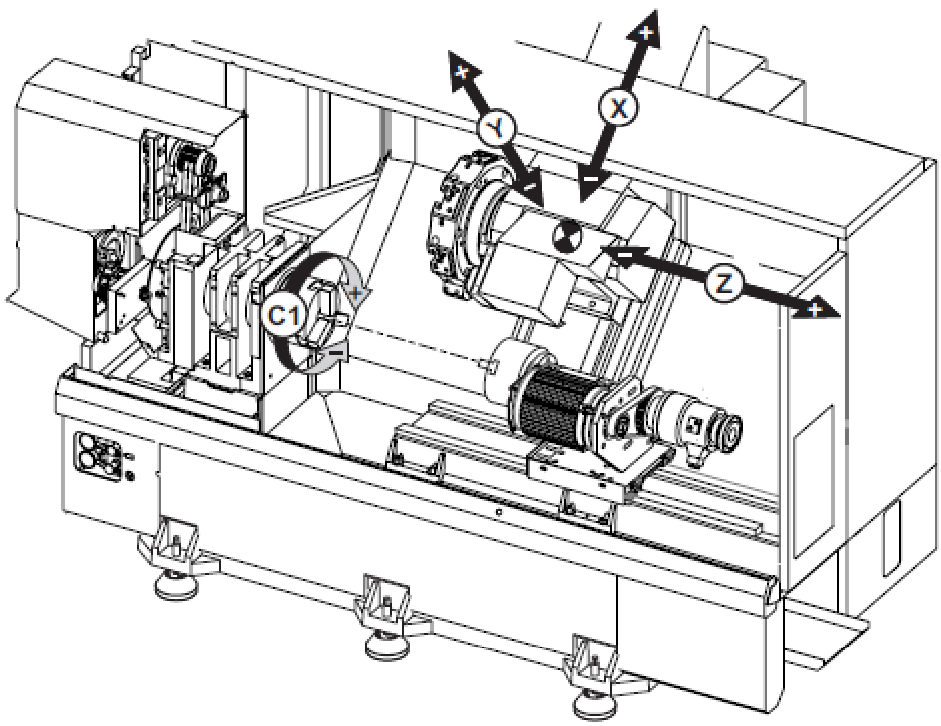
```
O00102 ( Y AXIS AND FACE DRILLING EX )
( MACHINE WITH G58 )
( MILL SINGLE END )
G00 G40 G99
G53 G00 X0.
G53 G00 Z-5.
N9 M01 ( .75 DIA. 4FL. E.M. ) ( ER-32 RADIAL HOLDER )
G53 G00 X0.
G53 G00 Z-5.
T909 ( .75 DIA. E.M. )
M154 ( ENGAGE C-AXIS )
G28 H0
G00 C0
M133 P611  (LIVE TOOLING FORWARD)
G98
G58 G00 Z.287 C0.
G58 G00 X2.5
G58 G00 Y-1.0
( C0.0 )
G01 X2.2 F4.8
G01 Z-.125
M14 (BRAKE ON)
G01 Y1.00 F4.8
G00 X2.6
( C180.0 )
G00 Y-1.0 C180.
G01 X2.2 F4.9.
M14 (BRAKE ON)
G01 Y1.0 F4.9.
G00 X3.5 Z2. M09
G28 Y0. C0.
G99 M135
G53 G00 X0.
G53 G00 Z-5.
M01

T101 (1/4” DRILL)
G54
G00 X1.2 Z0.1
G18 (X-Y PLANE)
G98
M154 (C-Axis engage)
C0
M133 P1833 (Live Tooling Drive Forward)
G81 Z-.575 R.1 F5.5
C60.
C120.
C180.
```

```
C240.
C300.
G00 G80 Z0.1
M155 (C-Axis Disengage)
M135 (LIVE TOOL DRIVE STOP)
G53 X0
G53 Z0
M30
%
```

Section II – DS (Dual Spindle) Series

Added 5/1/12

Section II - Dual Spindle Lathes (DS Series)

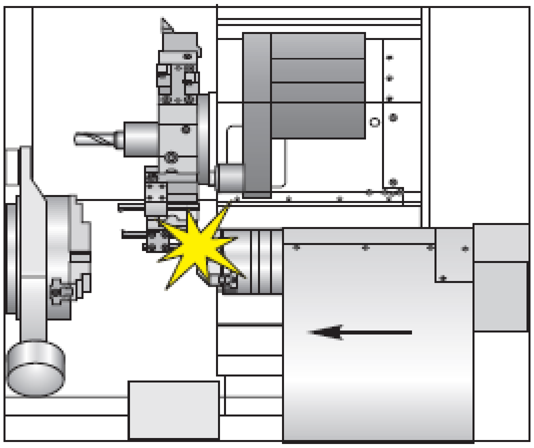


Dual-spindle lathe shown with optional Y-axis

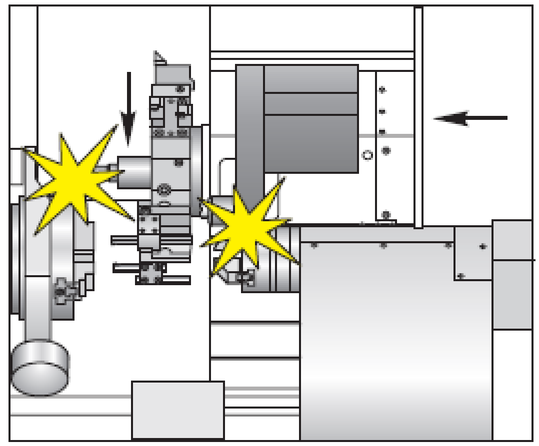
The figure above shows a DS-30Y which is a lathe with two spindles with an added Y-axis. The spindle on the left is referred to as the main spindle while the spindle on the right is referred to as a secondary or sub-spindle. Note the different axis of movement and their positive directions. The secondary or sub-spindle replaces the typical tailstock normally found on ST series lathes. Its axis of movement is the B axis with positive direction toward the end of the lathe next to the chip conveyer.

When you add another spindle and live tooling **clearance** of tools from crashing into one of the jaws of the two spindles becomes a major issue. Because of the various clearance problems always take the machine to a safe index position in X-axis first and then move the Z-axis second. When machining on the main spindle the sub-spindle needs to be moved back close to home position in the B axis.

Clearance Problems with Sub-Spindles and Live Tooling

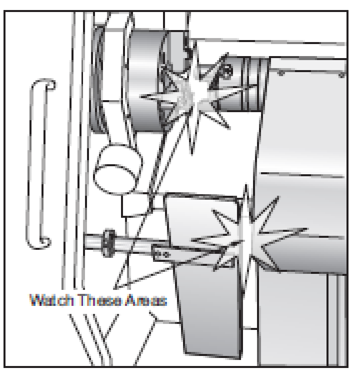


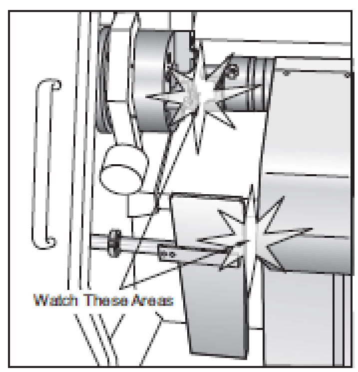
Long Sub-Spindle Tools



Live Tooling

When setting up and programming the DS Series Lathes one needs to be observing not only if the tool will clear the main spindle but live tooling or long tools in adjacent turret positions. On top of that one needs to be observing if main body or any tooling will clear the sub-spindle.

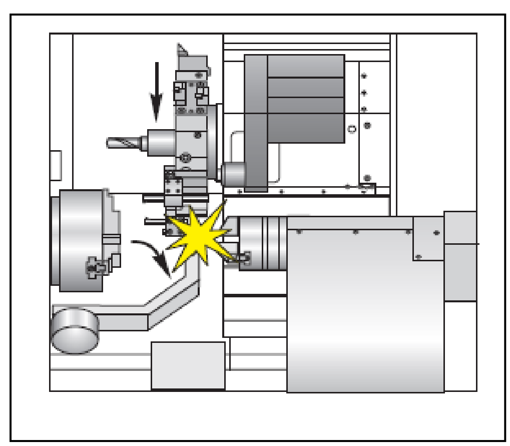




Also if the machine has a part catcher spindles could interfere with one another.

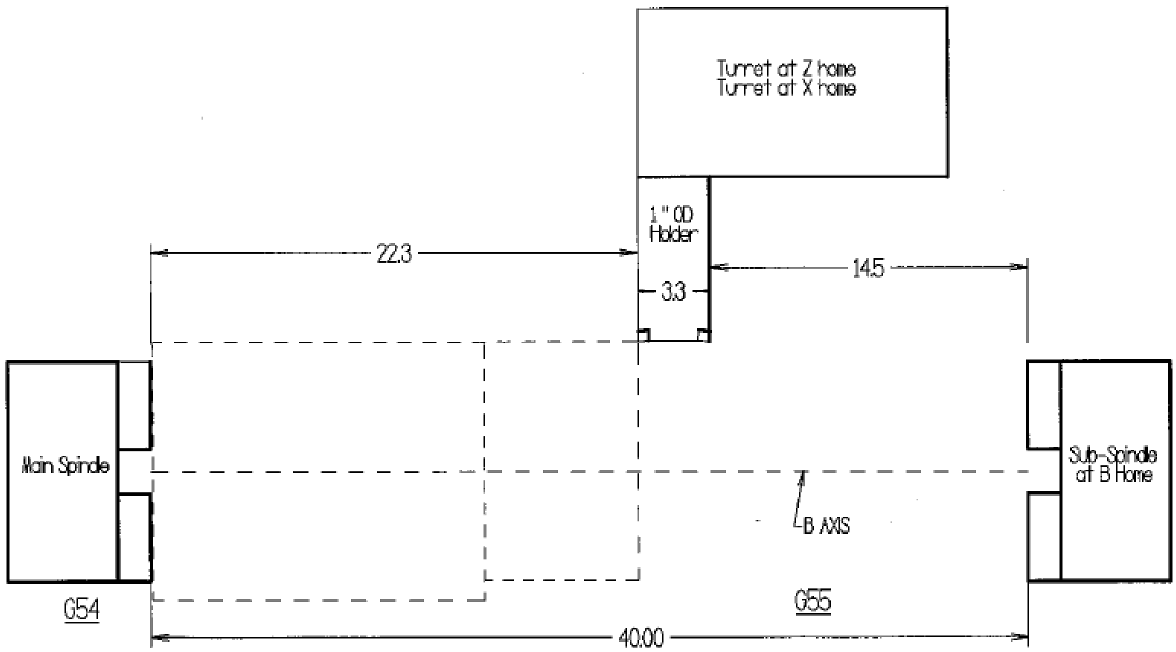
Care must be taken when using the Tool Presetter.

Make sure the sub-spindle and the turret are out of the way before lowering the Tool Presetter. Also make sure the longer tools or live tools will not hit the Tool Presetter when indexing to the next tool to be set.



Work Envelopes of the DS30 Series

(NOTE: Approximate values taken from Haas DS-30Y machine views from Haas web site.)



The above gives the cutting envelope of a Haas 20-5299 1.0 OD Holder. Other tooling will have different cut envelopes but this is a typical tool that would be used to turn the sub-spindle.

Note that the sub-spindle is at home position (B0). Note that the approximate distance from the face of the jaws on the main spindle and the face of the jaws on the sub-spindle is about 40". This is an approximate value and will vary depending on the thickness of the jaws attached to each spindle. This distance is easy to determine with the turret at home position.

In the jog mode press B button and press the jog mode button. Then the sub-spindle is easily moved forward. The sub-spindle may be brought forward just until the jaws touch the main spindle and the negative B value noted.

Also note that the sub-spindle must be brought forward for the tools in the turret to reach up to the jaws of the sub-spindle. In the case above the negative B value would be B-14.5. It is prudent to move the sub-spindle to B0 or home when the machine is doing work on the main spindle. This will prevent the turret from accidently crashing into the sub-spindle. When the sub-spindle is being worked on it must be brought forward or in the case above to a value of B-14.5. Then tools in the turret will be to be able to work up to the jaws of the sub-spindle.

Setting the DSL Work Offsets

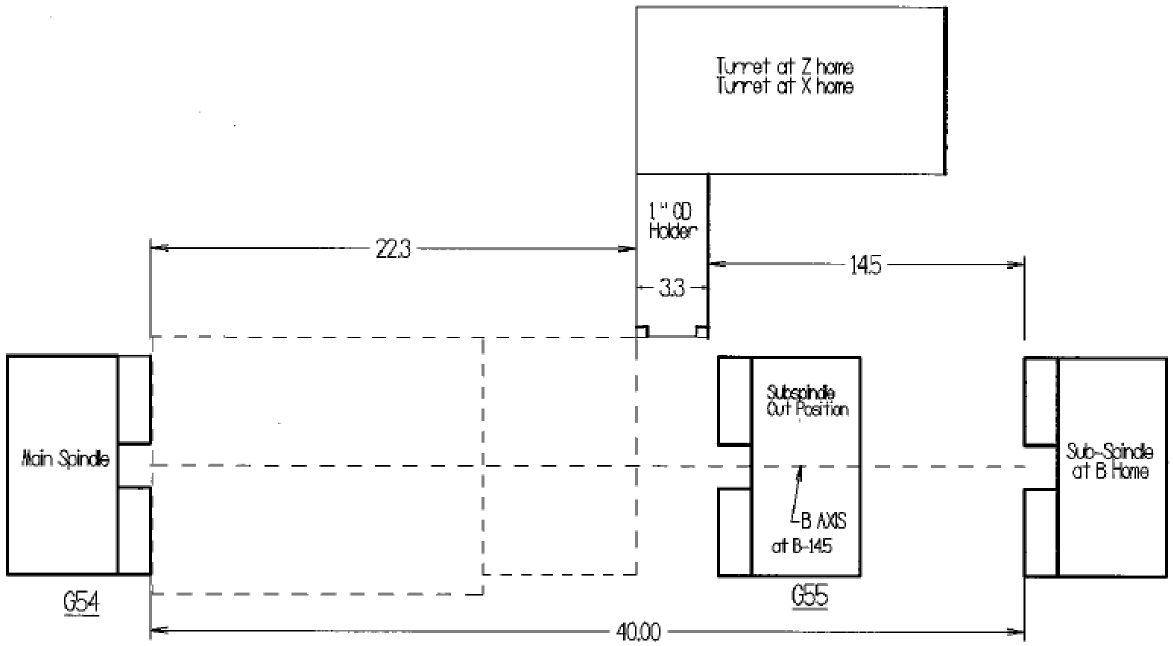


Illustration above shows positions of the turrets where machining will take place. Most users by convention will designate the main spindle G54 and the sub-spindle as G55. The G54 work offset will work in a similar fashion as other lathes. All tool geometries are determined using the tool pre-setter on the machine. Then work offset **G54** is determined by activating one tool and touching off on the face of the part in the main spindle and pressing the Z Face measure button.

With the sub-spindle **first** the sub-spindle is moved up to the cut or work position. In the example above the B-axis is manually moved to B-14.5. That position is noted and entered in the **G55** work offset B position column. For the above example B-14.5 is entered in the B column of **G55**. In this situation a line command of **G55 B0** will bring the B-axis position to B-14.5 (the cutting position for the sub-spindle).

Next one tool is activated, touched off on the face of the part in the sub-spindle and the Z Face Measure button depressed. Depending on how far the part is sticking out from the spindles the G54 Z values should be relatively small negative values (-4 to -6) On the other hand the G55 work offset Z values will be relatively large positive values around (+16 to +20).

Note: The B address character is used to call out absolute position along the sub-spindle axis. The units are in inches or millimeters with 4 or 3 fractional values respectively. Decimal points are required or the last digit will be interpreted as 1/10000 inch or 1/1000 millimeters.

Programming the Double Spindle Lathe

G14: Sub-spindle mode

Programming tool paths on the main spindle is done exactly as in any other Haas lathe. The only exception is that before machining the main spindle the sub-spindle should be placed at the B axis home position. This may be done with a simple G53 B0 command.

Programming the sub-spindle is made simple by using the G14 Mirror Image (Secondary Spindle Swap) command. The G14 command causes the sub-spindle to become the primary spindle. With the mirror image or G14 command the sub-spindle reacts to commands normally used on the main spindle.   M03, M04, M05 M19 will affect the sub-spindle as a mirror image of the main spindle. G50 limits the sub-spindle speed. G96 will set the spindle speed.

A G14 command will automatically mirror the Z-axis.   G15 cancels G14, as will M30, reaching the end of a program or by pressing Reset.

G41 and G42 work when the G14 is active just like when used when programming the main spindle. Haas recommends a G40 (cancel G41 or G42) command be placed in the block preceding the G14 command. Also G40 should be used before cancelling G14 with G15.

G184 sub-spindle Rigid Tap Cycle: G184 is programmed the same way as G84. G84 will not work on the sub-spindle with G14 active.

Canned cycles except will work on the sub-spindle with G14 code active. They must be prefaced with a G14 command.

Sub-spindle specific codes

M110: Sub-spindle Chuck Clamp

M111: Sub-spindle Chuck Unclamp

These codes open and close the sub-spindle chuck. Outside diameter versus inside diameter clamping of the sub-spindle is set with Setting 122.

G15 Specific codes (Main Spindle Active)

The following codes will only work when the active spindle is the main spindle.   If the active code is the sub-spindle (G14) these codes will cause an alarm. Use the M and G codes you normally would use on the main spindle for the sub-spindle with G 14 active.

M119 Orients the sub-spindle to a specific fixed position. If used in conjunction with a P value the sub-spindle will orient to a specific angle, the units degrees. Also an R value may be used to represent degrees with up to four places to the right of the decimal.

Example:

M119 P90 will orient sub-spindle to 90 degrees. M119 R123.4567 will orient the sub-spindle to 123.4567 degrees. Note the sub-spindle has no brake. Live tool machining needs to be done selectively.

M143 Spindle Forward. Starts sub-spindle movement in clockwise direction with your frame of reference being behind the sub-spindle. Use P to donate the sub-spindle speed.

M144 Spindle Reverse. Starts sub-spindle in the counterclockwise direction. This is the code used when swapping parts. It starts the sub-spindle going the same direction as a M03 command on the main spindle when G15 is active (Main spindle mode active).

M145 Spindle Stop. If a P value of zero is given the sub-spindle will coast to a stop. If no P value is indicated the spindle will decelerate to a stop.

Part Pass from Main Spindle to Sub-Spindle: The advantage of having a Dual Spindle lathe is that work on both sides of a part may be accomplished in one machine and one operation.

The following program is an example of a simple part pass-off from one spindle to the other without the spindles moving.

```
MACHINE FIRST SIDE COMPLETE
G53 G00 B-9.5 X0 (TAKE TURRET TO SAFE INDEX POSITION)
M01
N1000 (PART TRANSFER)
G103 P1 (LIMIT BLOCK LOOK AHEAD TO 1)
G53 G00 X0
G54 G00 Z0.2 (MOVE TURRET TO SAFE PART PASS POSITION)
M05 (STOP SPINDLE)
M145 (STOP SUB-SPINDLE)
G50 S500
G15 (MAIN SP PRIMARY)
M111 (SUB CHUCK OPEN)
G00 B-37.25 (RAPID POINT SUB-SPINDLE TRANSFER IN FRONT OF PART)
M12 (AIR ON)
G98 G01 B-38.742 F15. (MOVE SUB-SPINDLE TO PICKOFF POINT)
M110 (SUB CHUCK CLOSE)
G04 P1.
M13 (AIR OFF)
M11 (MAIN CHUCK OPEN)
G04 P2.
G55 G00 B0 (MOVE SUB-SPINDLE TO CUTTING POSITION)
```

```
M145 (STOP SUB-SPINDLE)
G99 (INCH PER REVOLUTION)
G103
M01
N2000 (2ND SIDE)
G55 G99
G53 G00 X0 Y0
G53 G00 Z-13.
G55 G00 B0 (MOVE SUB INTO CUTTING POSITION)
G14 (MACHINE SUB SIDE)
```

G199 Engage Synchronous Spindle Control

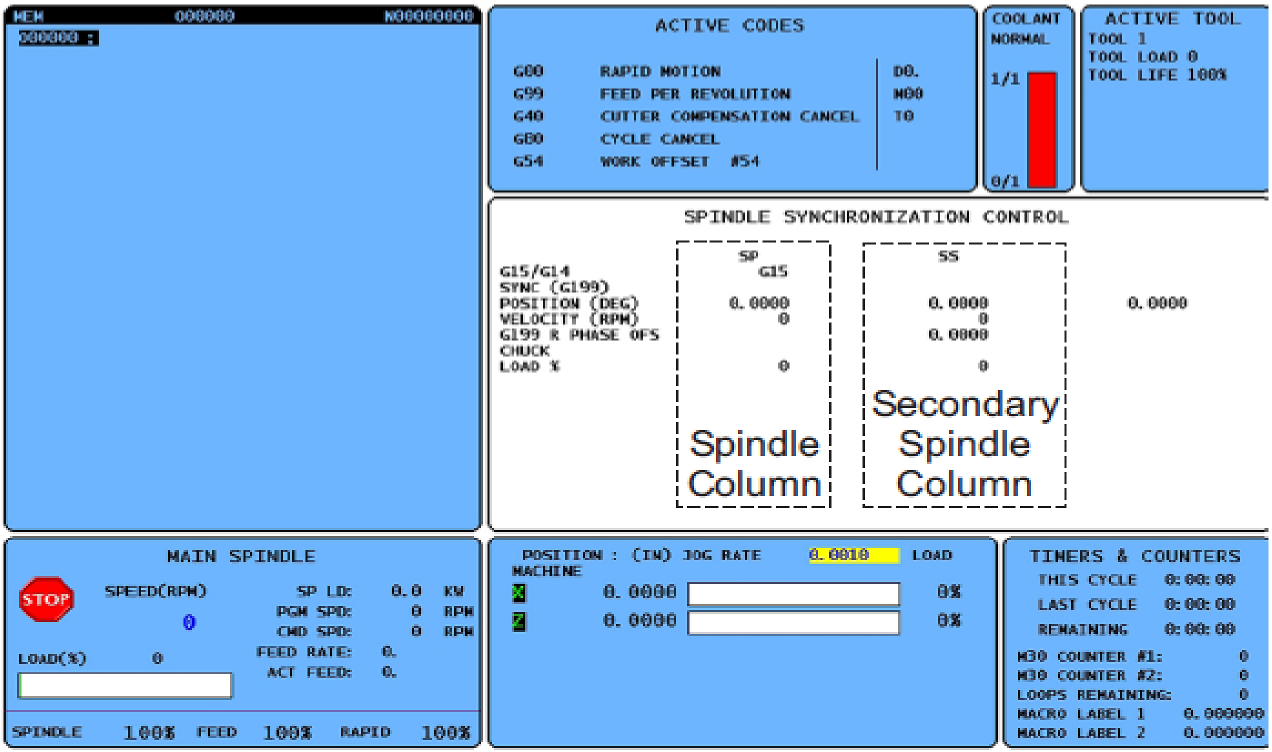
This code synchronizes the RPM of the two spindles. Dual spindle lathes have the ability to synchronize the spindle speeds of both spindles so cut off operations may be used with bar feeders. Position and speed controls are determined using the primary spindle. Speed commands to secondary are ignored. Spindles will remain synchronized until a G198 is called. During synchronous control both spindles will accelerate, maintain a constant speed and decelerate together. This prevents the spindle motors from fighting each other to maintain spindle speed.

R may be used to position the sub-spindle to a specified angle with respect to the main spindle.
G199 R30. will position the sub-spindle +30 degrees from the main spindles origin or C0.

G198 Disengage Synchronous Spindle Control

Disengages G199 synchronous control allowing independent control of the main spindle and sub-spindle speeds.

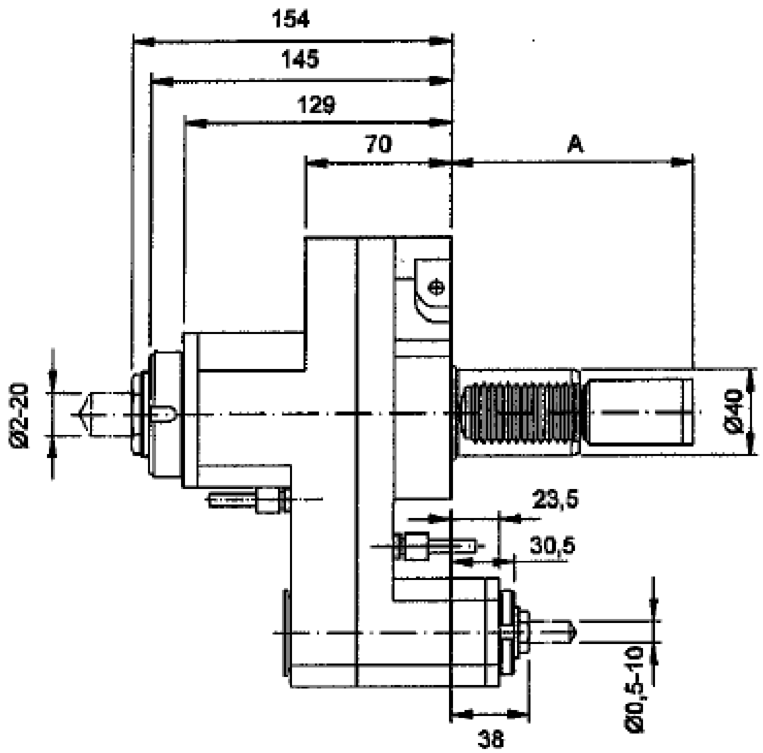
Synchronized Control Display



The screen above may be accessed by pressing the CURNT MOMDS button then press the (Page Up) button. The SP column gives the status of the main spindle, the SS column the status of the sub-spindle. The SYNC(G199) ROW indicates if G199 is active by appearing in the row. The POSITION (DEG) row gives the position of the main spindle and the sub-spindle. The third column gives the difference in degrees between the two. When the two are the same a zero appears in the third column. If the value is negative it indicates how much the sub-spindle is lagging behind the main spindle. This may be corrected by adding that amount to the C-axis value in the G55 work offsets. If the value is positive it shows how much the sub-spindle is ahead of the main spindle. This may be corrected by subtracting the amount in degrees from the C-axis value in the G55 offset. The final row gives the R value if it is programmed with the G199 code. If the two spindles are aligned properly the R values will be the same in the SP and the SS columns. This becomes important if features on one side must be aligned to features on the second side of a part.

The program on the following page is an example of a part pass-off from one spindle to the other with both the spindles moving.

```
N200 (PASS OFF)(PARAMETER 248 =550) (MAKE SURE THAT PAR 57 “SPINDLE NOWAIT”=1)
G103 P1 (LIMIT BLOCK LOOK AHEAD);
;
G00 G53 X0 Z-5. (SAFE INDEX POSITION)
G28 B0
T700
G50 S500
G97 S400 M03
G15 (MAIN SPINDLE PRIMARY)
G54 (FRONT SIDE)
M111 (SS CHUCK OPEN)
M144 P400 (SS M4 AT 400 RPM)
G199 (SPINDLE SINCHRONIZAZTION ON)
G98 (INCH/MIN)
G00 B-37.25 (RAPID POINT TO PART FRONT)
M12 (AIR BLAST ON, IF EQUIPPED)
G98 G01 B-38.742 F15. (PICKOFF POINT)
M110 (SS CHUCK CLOSE)
G04 P1. (NECCESARY FOR PROPER TIMING)

(INSERT CUT-OFF PROGRAM IF NECESSARY)
(THIS MAY REQUIRE A PART PULL)

M11 ( MAIN CHUCK OPEN )
G04 P1.(NECCESARY FOR PROPER TIMING)
G55 G00 B0 (RAPID BACK TO MACHINING POSITION)
G198(CANCEL SPINDLE SYNCH)
M10 (MAIN SPINDLE CLOSE)
M13 (AIR BLAST OFF)
G99 (BACK TO IPR)
Q103 P0 (NOT LIMITING, LOOK AHEAD)
M01
```



Ref. No.*	Ratio	n Max.	Nm Max.	Collet System	Coupling Length	A
4.200.810	1:1	4,000	40	ER 16-ER 32	Medium	117.55

*** Available for machines with sub spindle only.**

Above is live tooling designed for Haas Double Spindle Lathes from Eppinger.

With this tool live machining on the sub-spindle is possible. When in the G14 Sub-spindle mode live tooling may be programmed just as it is with the main spindle. Just remember that there is no spindle break for the sub-spindle for heavy cuts and drilling.