Haas Technical Publications

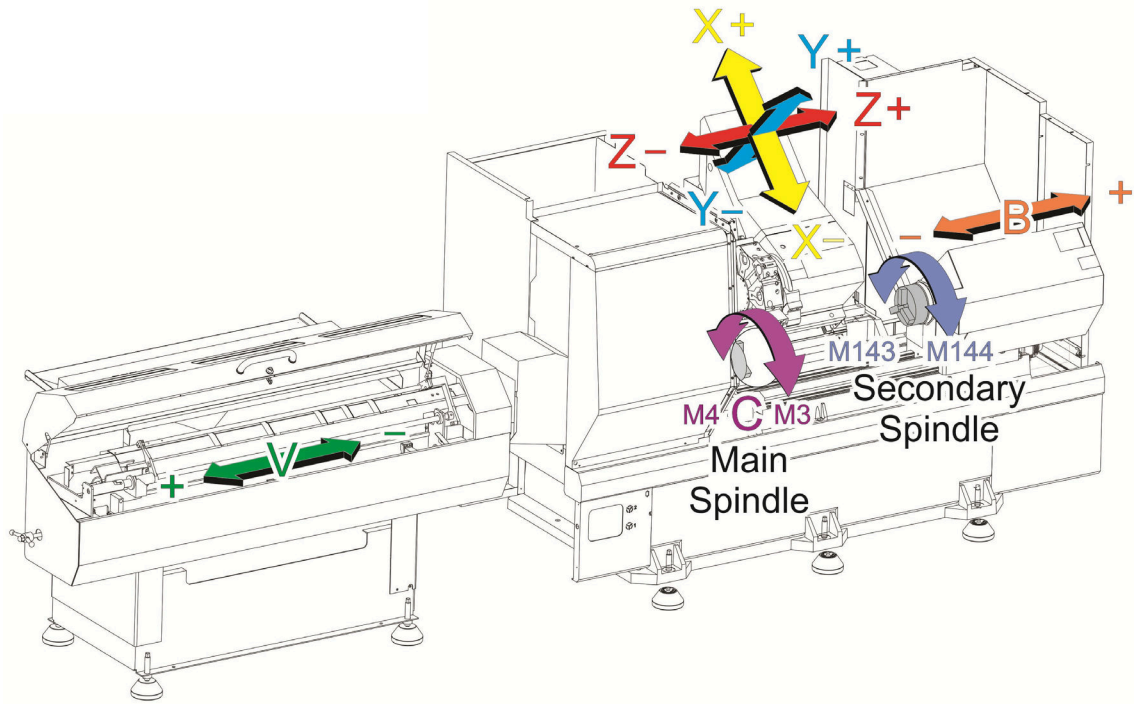# Y-AXIS LATHE APPLICATIONS TRAINING

Live-Tooling, Dual-Spindle, Y-Axis, Bar Feeder

This page intentionally left blank.

Contents

INTRODUCTION

This training document is intended to be used by Certified Haas Application's Engineers.

This document lays the framework for a 2-day Live-Tool Lathe Applications class. Chapters 1-8 to be covered on day one, and chapters 9-12 to be covered on the second day.

The information given builds on what you have already learned from the Haas Lathe and Mill Manuals. The reader must already have a basic knowledge of Haas G and M Code programming for both mills and lathes to get the most out of this document.

This document's purpose is to give the reader a working knowledge of the unique codes and practices needed to operate a Haas Lathe with the Live-Tooling, Dual-Spindle, Y-Axis or Bar Feeder options.

Software Version L11.10A was used. Upcoming software changes may render some of these notes obsolete.

The latest lathe manuals are full of new and useful information on Y-Axis lathes. Be sure to look at these, and have your customers download the latest versions.

New Machine Layout Drawings are also available on the Haas Website.

This page intentionally left blank.

SECTION 1.   HOMING THE MACHINE

POWER-UP RESTART

All of our **ST** lathes can be Zero Returned after Power On with a single button press, POWER UP RESTART.

Our Dual-Spindle and Toolroom Lathes (**DS**, **TL** models) require the operator to home each axis individually. This was added as a safety feature, to lessen the chance of a collision.

If POWER UP RESTART is pressed, on a DS or TL lathe, "FUNCTION LOCKED" will be displayed.

Check for Turret / Part interference, and make sure that one or both of the chucks is unclamped (DS only), and then ZERO RET>SINGL each axis, in the following order:

DS Machines: **B Y X Z A C V**

The B-Axis is the Tailstock.

The A-Axis is the Turret. Homing the A-Axis will bring tool T1 into cutting position.

The V-Axis is the optional bar feeder.

Note that (5) Axes can be viewed from the Position Screen at any one time.

Haas Technical Publications

To change which (5) Axes are displayed, we will:

Press the POSIT key

Press the F2 key

Select up to 5 axes you would like to display, by highlighting them and then pressing the WRITE ENTER key

Press F2 when finished

Note that the V-Axis is now visible on this DS machine

> **G28, G53**

HOMING THE MACHINE

You must always home the Y-Axis before the X. This must be done whether you are Powering-Up the machine, or if you command the X axis to the home position in a program. The next section, Machine Axes, gives more detail on this. See the following example.

G53 Y0.0
G53 X0.0

Note that the lathe will automatically return the Y-Axis to the home position after any tool change.

On a **Mill**, G90 is an Absolute move, and G91 is an Incremental move (see mill manual). On a **Lathe**, XYZ are used for Absolute moves and UVW are used for Incremental moves.

You can send the Y-Axis to the home position by entering G53 Y0.0. A G28 followed by a U or W will home the X or Z-Axis, but the use of a G28 V0. is not currently permitted. To keep things simple for the customer, showing them how to use a G53 for all axis-home movements is advised.

Because the V-Axis is used as both our bar feeder Axis and for incremental Y-Axis movement, a special code is needed to let the control know which axis we intended to move. A G160 command, prior to a V move, lets the control know that the move is for the bar feeder, and not an incremental move of the Y-Axis. When the move is complete, a G161 must be called.

To move the bar feeder V-Axis 1" to the right of its Home Position (Absolute Move), we would command:

G160
G00 V-1.0
G161

Note that a G00 V-1.0 command, without the G160, would move the Y-Axis incrementally -1.0.

To move the bar feeder V-Axis 1" to the right of its present position (Incremental Move), we would command:

G105 J1.0

The Current Bar Length, variable 3110, is updated when the above G105 J1.0 command is called. Commanding the bar feeder V-Axis directly (G00 V1.0), outside of the G105 command, will not update the Current Bar Length (variable 3110).

This page intentionally left blank.

SECTION 2.   MACHINE AXES, PLANES DEFINED

DS-30SSY
SERVO BAR 300
(BAR 3010SS)
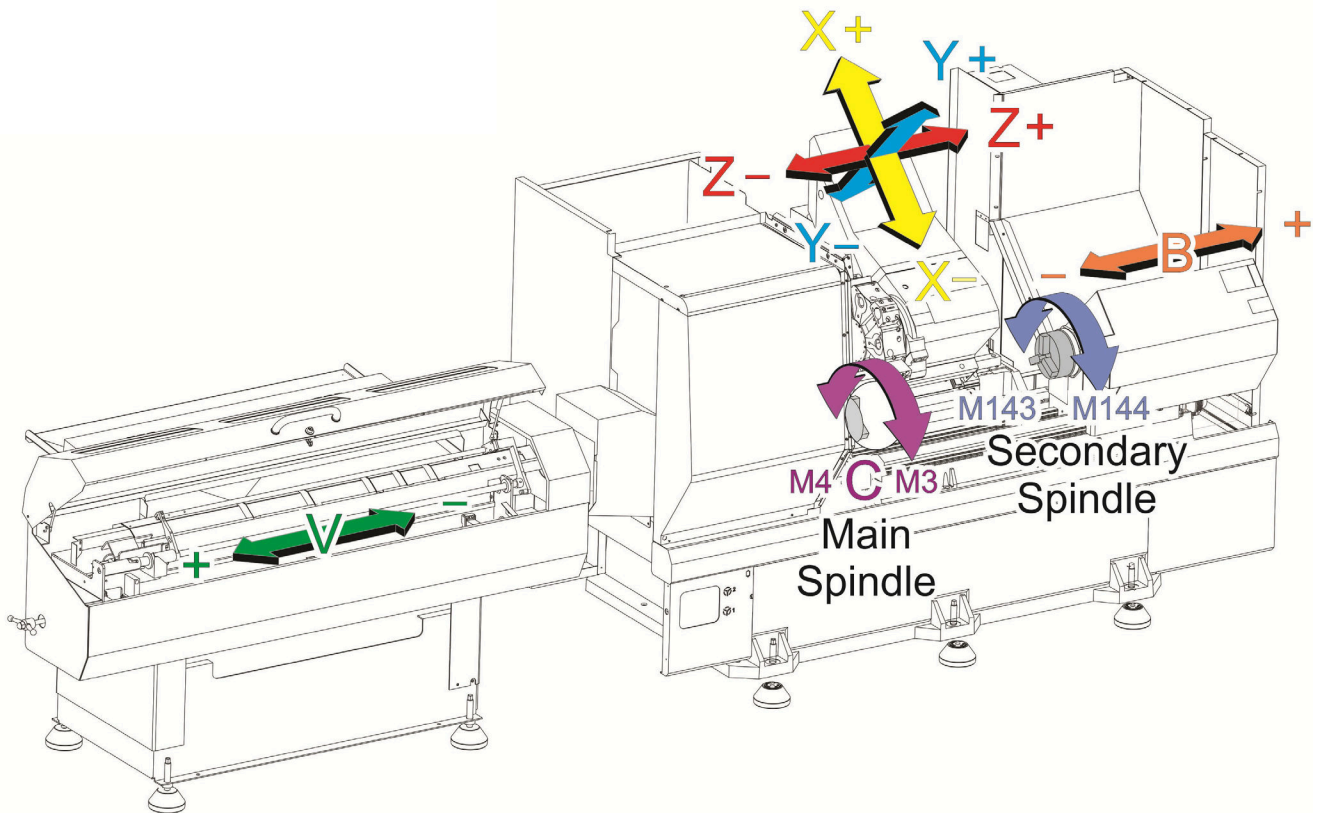
Not all of the axes shown are available on every machine.

AXES (XYZABCV), TRAVEL LIMITS

X-Axis

All X positions are programmed as diameters. This applies to milling, as well as turning. The top of a 1.0" diameter bar in the chuck would be X1.0, the bottom of that bar would be X-1.0.

Incremental X-Axis moves are commanded with a U move. These are also in diameter.

If you command "G00 U-1.0" in MDI, the turret will move incrementally towards the spindle by .5" which is -1.0" diameter.

The only X moves that are not defined as diameters are arc center-points and radius. These are our "I" and "R" values in G02/G03 moves. We would use the actual size of the radius, not multiplying the "I" by 2, as if it were a diameter. See the Live-Tool Milling section for more information and examples.

X-Axis Travel Limits

Each machine has different X-Axis travel limits. This is important when deciding what approach to take when live-tool milling. If you are axial-milling with a .5" diameter endmill, you will need to travel to X-1.5 in order to mill a 1" boss. This may work well on an ST-20SS, but a standard ST-20 would not have the travel to reach that far below the X centerline. In that instance we would program the boss using XC moves, or a G112 cycle.

Model Number	Travel Below X CL, Axial Live Tool	TS Morse Taper Size
OL-1	6.00	-
ST-10	0.35	3
ST-10Y	0.35	3
ST-20	0.40	4
ST-20Y	0.40	4
ST-20SS	1.70	4
ST-20SSY	1.70	4
ST-30	0.48	4
ST-30Y	0.48	4
ST-30SS	1.62	4
ST-30SSY	1.62	4
ST-40	1.20	5
ST-40L	1.20	5
DS-30	0.48	-
DS-30Y	0.48	-
DS-30SS	1.62	-
DS-30SSY	1.62	-

DS-30SSY Shown

Values shown are actual inches below X centerline, not diameter values

Y-Axis

The Y-Axis on lathes moves in the opposite direction of what you may be used to on your mills. The positive direction is towards the operator, the negative is away. Y-Axis moves are programmed radially, unlike the X-Axis which is programmed with diameter values.
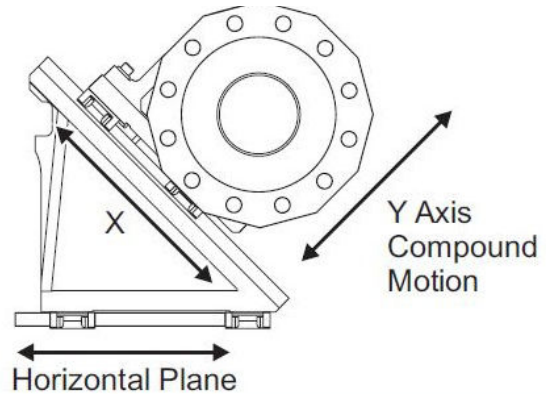
Incremental Y-axis moves are commanded with a V.

If you command "G00 V-1.0" in MDI, the turret will move incrementally away from the operator by 1.0"

When milling in the G19 (YZ) plane, or turning in the G18 (XZ) plane, you can program an arc with a "J" or an "R". See the Live-Tool Milling section for more information.

Y-Axis Travel Limits

Each Haas Y-Axis lathe has +/- 2.00" of travel.

There is a narrow corner of the work envelope, where a positive Y-Axis movement is constrained, as the X-Axis reaches its physical limit. This area is marked as "B" in the diagram to the right. This is why we require the Y-Axis to be homed first, before the X-Axis.

The following command would not be executed, because the coordinates fall outside of the machine's travel limits:

G53 G00 Y2.0

G53 G00 X0.0

Haas Technical Publications
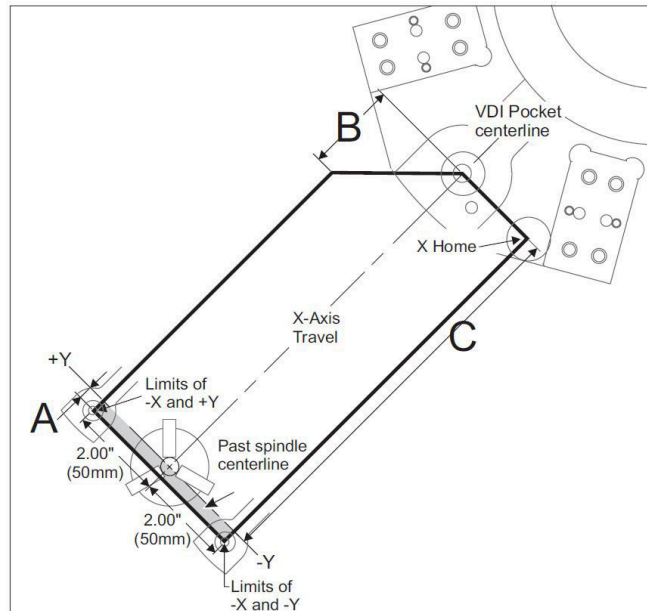
Z-Axis

The Z-Axis moves the turret away from the Main Spindle (Z+), or towards it (Z-).

Incremental Z-Axis moves are commanded with a W.

If you command "G00 W-1.0" in MDI, the turret will move incrementally towards the Main Spindle by 1.0".

SECTION 3.   SPINDLE DIRECTION, SPEED

	Forward	Reverse	Speed Code	Tapping Speed
Main Spindle	M03	M04	S	S
Secondary	M143	M144	P	S
Live-Tool	M133	M134	P	S

The following will command the live-tool to spin in the forward direction, at 2000 rpm:

G97 M133 P2000

We need to be very careful when using right-angled or extended live-tooling holders. These often contain gearing that will turn the tool in the opposite direction of the drive. Test all live-tool holders to make sure the tool is turning in the intended direction before cutting a part.

You can command the Secondary Spindle in the same manner:

G97 M143 P2000

While we may have a need to rotate the Secondary Spindle in this way, all machining on the second spindle must use a G14 Secondary Spindle Swap command. See the Dual Spindle section for more information. A brief example is shown below:

G14 (Spindle Swap On)

G97 M03 S2000

G15 (Spindle Swap Off)

All tapping speeds are commanded with an S value on the Haas Lathe, regardless of whether we are using the Main Spindle, the Secondary Spindle, or Live-Tooling.

When commanding a spindle to rotate in the forward direction ⟳ (M03, M143, M133), it will appear to move counter-clockwise (CCW), from the operator's perspective.

Below is a visual aid showing the direction of the Main and Secondary Spindles on a DS machine, for turning (M3/M143), C-Axis rotation (M154), and Spindle Orientation (M19/M119).

From the illustration, we see that the following command will move the main spindle clockwise (from the perspective of the part face) by 90°

G54

M154

G00 C90. (ROTATES THE SPINDLE CLOCKWISE, IN OPERATOR VIEW)

DS-30SSY
SERVO BAR 300
(BAR 3010SS)

M3

C0.
OR
M19 P0.
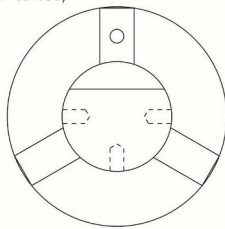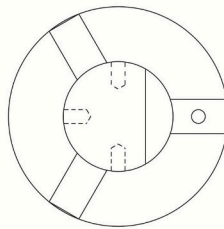
C90.
OR
M19 P90.
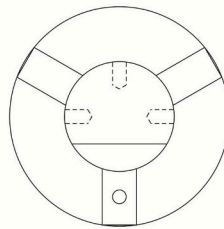
C180.
OR
M19 P180.
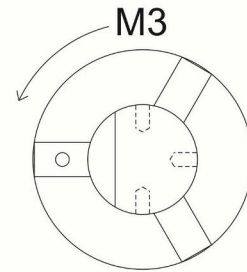
C270.
OR
M19 P270.
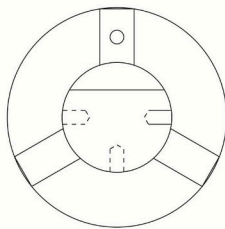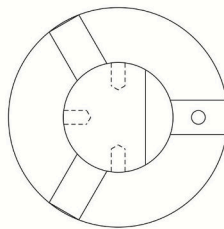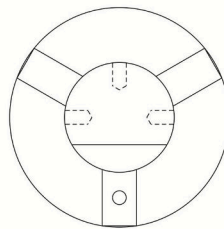
MAIN SPINDLE

M143

M119 P0.

M119 P90.

M119 P180.
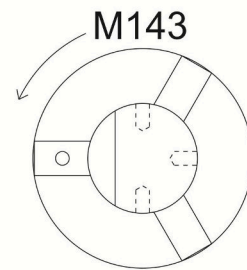
M119 P270.

SECONDARY SPINDLE
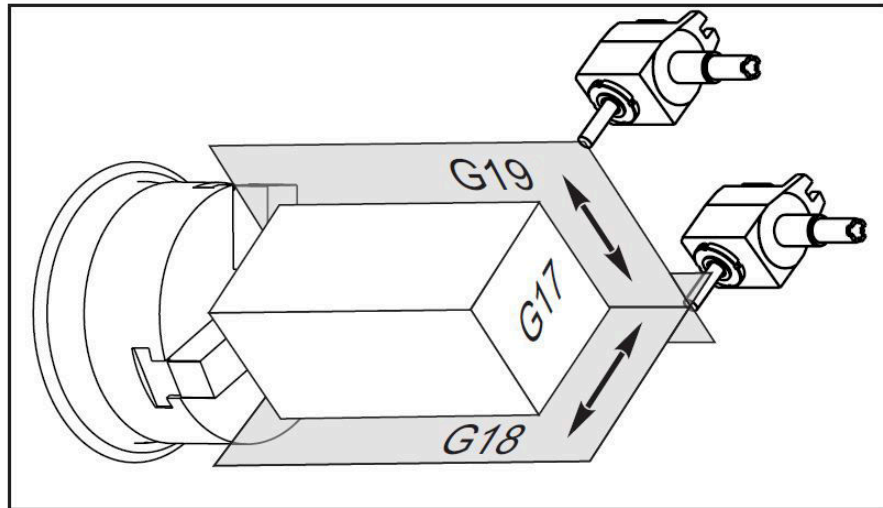
PLANES G17, G18, G19

We use planes (G17, G18, G19) on our Y-Axis lathes to tell the control which axes (XYZ) G02 and G03 circular motions are performed in, and which direction to move when using a canned cycle (depths, rapid planes).

Calling up the wrong plane can cause our arcs to become distorted and our canned cycles to drill in the wrong directions.

If we are programming straight lines, from one XYZ point to another, the plane selection will not have an effect on our part.

Note that there was a look-ahead issue in early L11.xx series software (prior to L11.06), where the control could change planes too soon, when another plane was called out later in your program.

The solution is to either:

1)   Stop look-ahead before your plane change by placing an M01, /, M00, or G103 P1;;; before any plane change (G17, G18, G19) or

2)   Upgrade your software to the latest L11.10A software or newer.

G17, XY Plane

G17 is used when creating arcs (G02, G03) in the XY plane, for live-tool axial milling.

Live-tool canned cycles are designed to run either axially (i.e. G83) or radially (G243). For this reason, all live-tool canned cycle can be run in the default **G18** plane. Do not run your axial live-tool canned cycles in the G17 plane.

It must be noted that the live-tooling canned cycle examples in Lathe Manual 96-8700 Rev AL show the use of the G17 plane for axial live-tool drilling cycles. This was incorrect. Have your customers download the latest lathe manual (Rev AN or newer) from the website.

See the sections for Live-Tool Canned Cycles and Live-Tool Milling for examples, and the use of planes.

G18, XZ Plane

G18 is used for creating arcs (G02, G03) in the XZ plane in our turning operations, and is the default plane for all of our Live-Tool Canned Cycles.

Live-tool canned cycles do not abide by the G17, G18, and G19 plane rules. Teach your customers to **use the G18 plane for all live-tool canned cycles (L11.02 and newer)**. This was done as a means to simplify all canned cycles and prevent collisions on our customer's machines.

See the sections for Live-Tool Canned Cycles and Live-Tool Milling for examples, and the use of planes.

G19, YZ Plane

G19 is used for creating arcs (G02, G03) in the YZ plane, for live-tool radial milling.

See the sections for Live-Tool Canned Cycles and Live-Tool Milling for examples, and the use of planes.
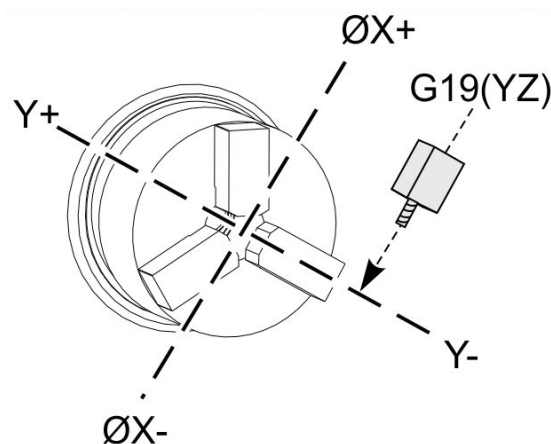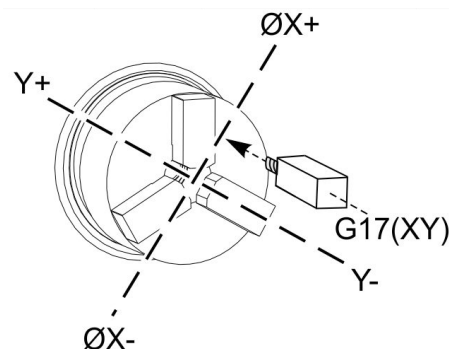
SECTION 4.   RESOURCES

ES DOCUMENTS

Listed below are important ES Documents that you need to be familiar with.

- ES0335 Lathe Tooling and Toolholding Information
- ES0603 Spindle Liners
- ES1003 Spindle, Chuck and Drawtube Specs
- ES0088 Spindle Torque Charts
- ES1001 Parts Catcher Option
- ES0761 Chuck Weights

The latest versions of these documents can be found on the Portal at :

PORTAL>Sales>ES Docs
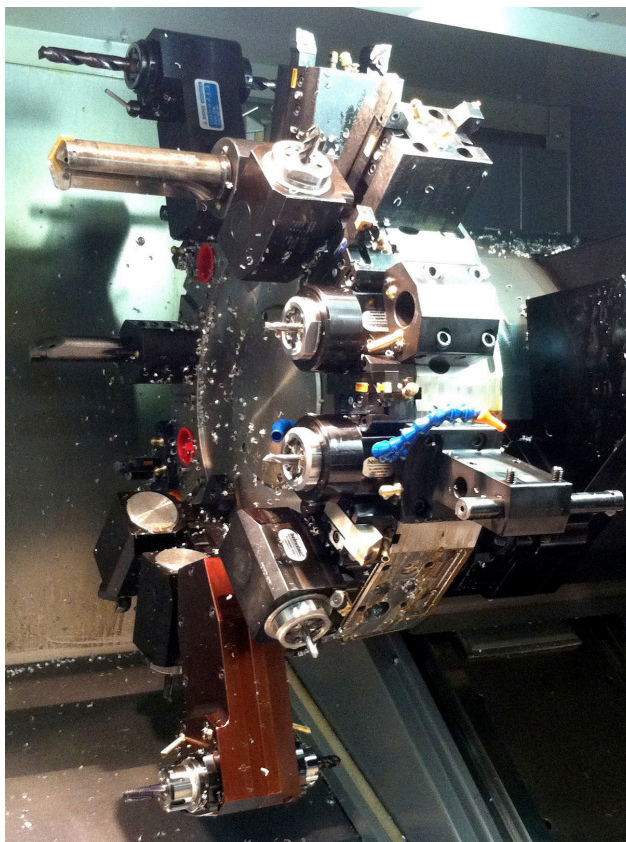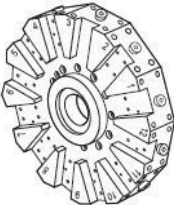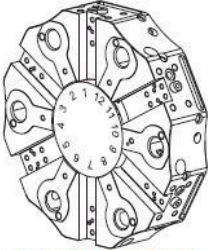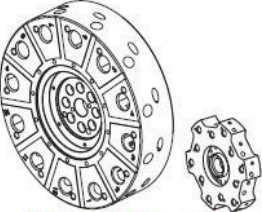
ES0335 Lathe Tooling and Toolholding Information.

This document gives us part numbers for lathe tooling, and helps define the differences between lathe turret designs.

Lathe Turret Visual Identification

Turret	Availability
12-Station BOT	**12-Station BOT** Standard on: ST-10 ST-20 ST-30 ST-40 Optional on: ST-10 HE ST-20 HE
12-Station Hybrid (VB)	**12-Station Hybrid** Standard on: ST-20Y ST-30Y DS-30 DS-30Y Optional on: ST-20 ST-30 ST-40
24-Station Hybrid (VB)	**24-Station Hybrid** Standard on: ST-20SS ST-20SSY ST-30SS ST-30SSY DS-30SS DS-30SSY Optional on: None
12-Station VDI ST-10 12-Station VDI	**12-Station VDI** Standard on: ST-20 HE Optional on: ST-20 ST-30 **ST-10 VDI** Standard on: ST-10Y ST-10 HE Optional on: ST-10
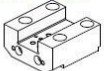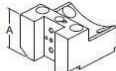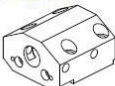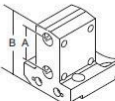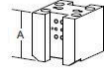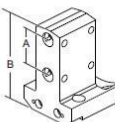
Haas Technical Publications
Lathe Tooling and Toolholding Information

VB24

3/4" (20 mm)
Available on SS Models Only
ST-20SS
ST-20SSY
ST-30SS
ST-30SSY
DS-30SS
DS-30SSY

BOLT-ON TOOLS PARTS LIST

Face Grooving	Part No.	Size	Parting Tool	Part No	
	BOT24FG-3/4 BOT24MFG-3/4	3/4" 20 mm		BOT24OD-3/4 BOT24MOD-20 Height(A): 2.3" (58.4 mm)	
Boring Bar	BOT24ID-1 BOT24MID-25	1" 25 mm	**Extended Parting Tool**	BOT24EXTOD- BOT24EXTMOD	
	BOT24ID-1_1/4 BOT24MID-32	1-1/4" 32 mm		Height (A): 4.55" (115.6 mm	
Twin Bore	BOT24ID-3/4T BOT24MID-20T	3/4" 20mm	**Extended Twin Turn**	BOT24EXTOD-3/4T BOT24EXTMOD-20T	3/4" 20 mm
	Dim. A: 2.15" btw. centers 54.6 mm			Height (A): 3.525" (89.5 mm)	
	Dim. B: 4.2" 106.7 mm		**Facing**	BOT24ODF-3/4 BOT24MODF-20	3/4" 20 mm
Extended Twin Bore	BOT24EXTID-3/4T BOT24EXTMID-20T Dim. A: 2.15" btw. centers 54.6 mm	3/4" 20mm			
	Dim. B: 5.325" 135.3 mm		**Tool Kits**	BOTKIT24 BOTKIT24M	Inch Metric

ES0335, REV T   JUN 2012   Page 10 of 16

Haas Technical Publications

ES0603 Lathe Spindle Liners

This document shows everything you need to choose and install spindle liners on your customer's specific machine.

Haas Technical Publications
Lathe Spindle Liners

Return to Top

Spindle Liner Kit Information

Extruded Spindle Liners

Lathe Model *Bar Capacity*	Adapter Kit	Liner Kit	Liner Kit Contains (*)	Fits Bar Diameter (inch)	Fits Bar Diameter (mm)
ST-10, ST-10Y 1.75" (44 mm)	LAGK100	LINERKIT-10	LINER375	.25-.375	6-10
			LINER500	.5	13
			LINER750	.75	19
			LINER875	.875	22
			LINER1000	1.0	25
			LINER1125	1.125	29
			LINER1250	1.25	32
			LINER1375	1.375	35
			LINER1500	1.5	38
ST-20, ST-20Y ST-20SS, ST-20SSY DS-30, DS-30Y DS-30SS, DS-30SSY 2.0" (51 mm)	LAGK200	LINERKIT-20	Includes LINERKIT-10, Plus:		
			(**) LINER1750-20	1.75	44
ST-20 with Big Bore ST-20Y with Big Bore 2.5" (64 mm)	LAGK250	LINERKIT-20B	Includes LINERKIT-10, Plus:		
			(**) LINER1750-20B	1.75	44
			LINER2000-20B	2.0	51
			LINER2250-20B	2.25	57
ST-30, ST-30Y 3.0" (76 mm)	LAGK300		Includes LINERKIT-10, Plus:		
ST-30SS, ST-30SSY DS-30 with DS-3B DS-30Y with DS-3B DS-30SS with DS-3BSS DS-30SSY with DS-3BSS 3.0" (76 mm)	LAGK300-89	LINERKIT-30	(**) LINER1750-30	1.75	44
			LINER2000-30	2.0	51
			LINER2250-30	2.25	57
			LINER2500-30	2.5	64
			LINER2750-30	2.75	70

(*) All extruded liners listed are available individually. Use the listed part number to order.
(**) Must use the 1-3/4" liner as a sleeve for smaller diameter liners. Refer to the installation instructions.

Kit LINER318S30BB

ST-30 with Big Bore ST-30Y with Big Bore 4.0" (102 mm)	N/A	LINER318S30BB	Part Description	Quantity
			PVC Spacer	10
			(***) Bar Guide Disk (Blank)	22
			Compression Nut	1
			Bar Guide Disk (1.0" I.D.)	11
			Liner Adapter .25" - 3.125" Dia.	1
			Liner Retainer Ring	1
			Liner Insert 318 S30 BB	1
			Liner Insert 300 S30 BB	1

(***) Order item UDK15 to receive 33 replacement bar guide disks. All other parts listed are available only in kit LINER318S30BB; they are not available individually.

ES0088 Spindle Torque Charts

This document shows the Torque Curves for the customer's machine spindles, and their live tooling. This will help you decide the optimal RPM to run a spindle for a certain task.

DS-30 (Main Spindle)

A2-6 - 4,000rpm - 30hp (Belt Drive)

Standard: DS-30
Optional: DS-3B (Chuck and draw tube size change only)
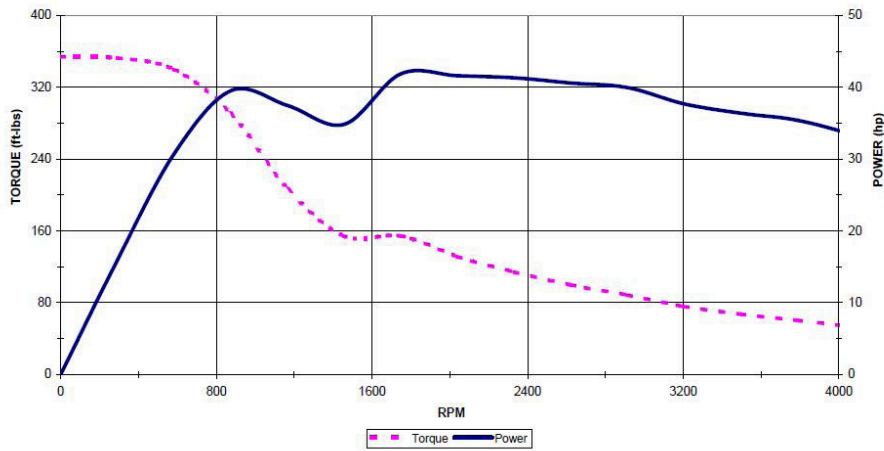
ES-0088 5/7/2012

Live Tooling

3,000rpm - 7.5hp / 12hp (Gear Drive)

Standard: Y-axis Lathes (7.5hp)
Optional: LT, LT2-HT, ST-10 through ST/DS-30 (7.5hp)
LT4, ST-40/ST-40L (12hp)
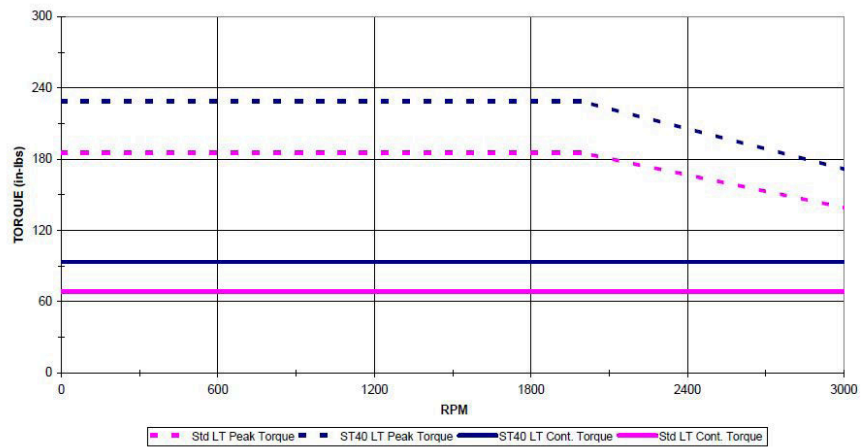
ES-0088 5/7/2012

Page 43

MACHINE LAYOUT DRAWINGS

Every customer with a Haas Y-Axis lathe should have, and be familiar with, the Machine Layout Drawing (MLD) for his machine.

These drawings give travel limits and other information, specific to each machine.

These drawings can be downloaded from the Haas website, by following the link shown for each machine.

3rd PARTY AFTERMARKET RESOURCES

Live-Tool Drive Units
- Exsys/Eppinger – http://www.exsys-tool.com
- Parlec – http://www.parlec.com
- Heimatec – http://www.heimatecinc.com
- Benz – http://www.benz-inc.com
- MD Tooling - http://mdtooling.com
- MTS Driven Tools - http://www.mtsdriventools.co.uk
- WTO - http://www.wto-usa.com
- Von Ruden - http://www.vonruden.com
- Velocity - http://www.velocityproducts.com
- Holdwell - http://www.holdwell.com
- Global CNC - http://www.globalcnc.com

Steady Rests
- Kitagawa – http://www.kitagawa.com
- LMC Workholding – http://www.lmcworkholding.com
- LNS America – http://www.lns-america.com
- Rest In Peace – http://steadyrest.net
- Rohm Products – http://www.us.roehm.biz
- ATS Workholding – http://www.ats-s.com
- SMW Autoblok – http://www.smwautoblok.com

 There are several tooling options worth mentioning for those with DS machines.

1) Look at ES document ES0335, for the Extended Twin Bore and Extended Twin Turn bolt-on toolholders for the VB24 turret.

2) Extended reach axial double ER32 live-tooling is available from companies like BENZ (www.benz-inc.com).

3) Companies like Heimatec also make narrow stick tool holders that nest in between live-tooling heads. This increases the number of tools a turret can accommodate.

This page intentionally left blank.

SECTION 5.   LIVE-TOOL CANNED CYCLES

CHANGES BETWEEN VERSIONS

> Running L11.02 or newer?
> Use the **G18** plane for all of your **Axial and Radial** live-tool drilling/tapping/boring canned cycles.
> Use only designated Radial Canned Cycles (i.e., G241) for drilling/tapping/boring radially.

The Haas lathe uses different G codes for Axial and Radial Canned Cycles.

To Peck Drill axially (in the Z direction), we will use a G83. To Peck Drill radially (along the X-Axis), we will use a G243. Both the G83 (axial) and G243 (radial) Peck Drill Canned Cycles are called using the **G18** plane.

Prior to L11.06 it was possible to force a G83 (or similar) canned cycle to cut radially instead of axially by changing the plane from a G18 to a G17. Since L11.06 the control will ignore all plane call-ups, and will run Axial Cycles along the Z-Axis only and Radial Cycles along the X-Axis only.

Designated Radial Canned Cycles (G241-249, G195-196) were made usable in L11.02. These cycles will always cut along the X-Axis, regardless of plane selection.

Live-Tooling Lathes with software <u>older than L11.02</u> must use the default G18 plane to drill axially, and the G17 plane to drill Radially (O.D.).

Note that the examples shown in Lathe Manual 96-8700, Rev. AL, are incorrect when they show axial cycles using a G17.

Many changes have been made to our Lathe Manual recently. Be sure to have your customers download the latest revision (Rev. AN or newer).

Axial Cycles: G81, G82, G83, G85, G86, G87, G88, G89, G95, and G186

Radial Canned Cycles: G195, G196, G241, G242, G243, G245, and G249

AXIAL CANNED CYCLES
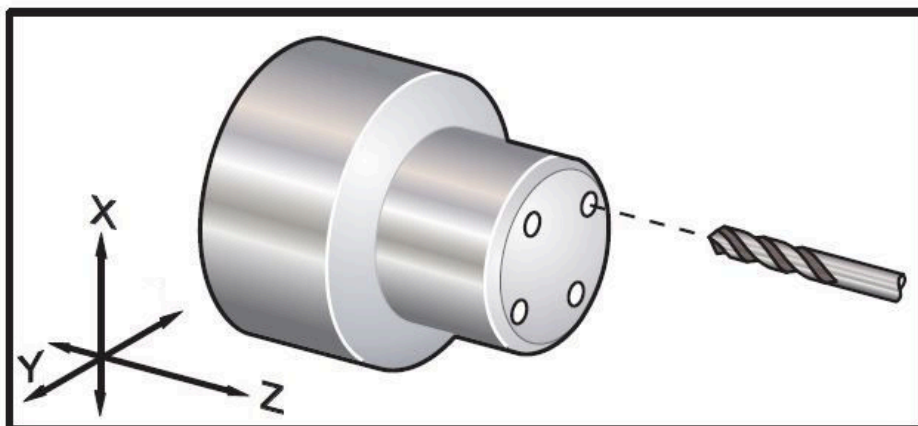
Axial Tool

Below are our Axial Canned Cycles, as listed in the manual (96-8700 Rev AN April):

Axial Cycles:

G81 Drill Canned Cycle (Group 09)
G82 Spot Drill Canned Cycle (Group 09)
G83 Normal Peck Drilling Canned Cycle (Group 09)
G85 Boring Canned Cycle (Group 09)
G86 Bore and Stop Canned Cycle (Group 09)
G87 Bore and Manual Retract Canned Cycle (Group 09)
G88 Bore and Dwell and Manual Retract Canned Cycle (Group 09)
G89 Bore and Dwell Canned Cycle (Group 09)
G95 Live Tooling Rigid Tap (Face) (Group 09)
G186 Reverse Live Tool Rig Tap (For Left Hand Threads) (Group 09)

Refer to Lathe Manual 96-8700, Revision AN or newer, for detailed descriptions of these cycles. That manual has been updated, and contains the latest information on these cycles.

In order to use Live Tool Canned Cycles, Parameter 315, Bit 1, NO SPIND CAN needs to be set to "1". There is more information on this, in the FAQ section of this document.

It is always best to use the M154 (C-Axis) to orientate the Main Spindle, as opposed to using an M19. This is a much more accurate and repeatable method. The Secondary Spindle on a DS machine has no C-Axis, and indexing on that spindle must be commanded with an M119, or a G14 M19. See the Dual Spindle section for more information.

The following axial drilling example will work on all software versions:

Axial Cycle Example:
```
%
O00810 (G81- AXIAL DRILLING)
T2323
G54 G18 G98(USE G18 PLANE)
M154 (Engage C-Axis)
G00 X6. C0. Z1.
G00 X2. Z0.25
G97 P1500 M133
G81 G98 X1. Z-1. F10.
C135. (DRILL SECOND HOLE AT C135.)
G00 G80 Z0.25
M155
M135
M09
G28 H0. (Unwind C-Axis)
G00 X6. Z1.
M30
%
```

Program examples for each of the Axial Canned Cycles, can be found at the end of this document, in Appendix A.

RADIAL CANNED CYCLES
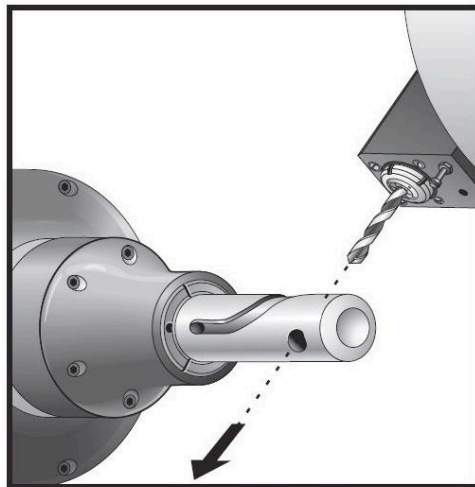
Radial Tool

Radial Canned Cycles:

G195 Forward Live Tool Radial Tapping (Diameter) (Group 00)
G196 Reverse Live Tool Radial Tapping (Diameter) (Group 00)
G241 Radial Drill Canned Cycle (Group 09)
G242 Radial Spot Drill Canned Cycle (Group 09)
G243 Radial Normal Peck Drilling Canned Cycle (Group 09)
G245 Radial Boring Canned Cycle (Group 09)
G249 Radial Bore and Dwell Canned Cycle (Group 09)

Refer to Lathe Manual 96-8700, Revision AN or newer, for detailed descriptions of these cycles. That manual has been updated, and contains the latest information on these cycles.

It is always best to use the M154 (C-Axis) to orientate the Main Spindle, as opposed to using an M19. This is a much more accurate and repeatable method. The Secondary Spindle on a DS machine has no C-Axis, and indexing on that spindle must be commanded with an M119, or a G14 M19. See the Dual Spindle section for more information.

The G195 radial tapping cycle is unique, and differs from other canned cycles. See the current manual (96-8700 Rev AN April or newer) for a full description.

The tool must be positioned to the Start Point before commanding G195/G196. This G code is called for each hole being tapped. The cycle begins from the current position, tapping to the X-Axis depth specified. An "R" plane is not used. Only X and F values should be used on G195/G196 lines. The tool must be positioned to the Start Point of any additional holes before commanding G195/G196 again.

Program Example

```
%
O01950 (LIVE TAP - RADIAL)
T101
M154                        (Engage C-Axis)
G00 G54 X6. C0. Y0. Z1.
G00 X3.25 Z-0.75 C0. Y0.    (Start Point)
G99                         (Must Set to Feed Per Rev. for this cycle)
S500
G195 X2. F0.05              (Taps to X2., bottom of hole)
G00 C180.                   (Index C-axis. New Start Point)
G195 X2. F0.05
G00 C270. Y-1. Z-1.         (Optional Y and Z-axis positioning, New Start Point)
G195 X2. F0.05
G00 G80 Z0.25
M135
M155
G00 G28 H0.                 (Returns C-Axis to Home Position)
G00 X6. Y0. Z3.
G98
M30
%
```

While not shown above, the G195 cycle can be used to Vector Tap along the X and Z axes. Vector tapping using the Y-Axis is not allowed.

Program examples for each of the Radial Canned Cycles, can be found at the end of this document, in Appendix B.

		Rapid
		Feed
	●	Start or End of stroke
	◇	Dwell
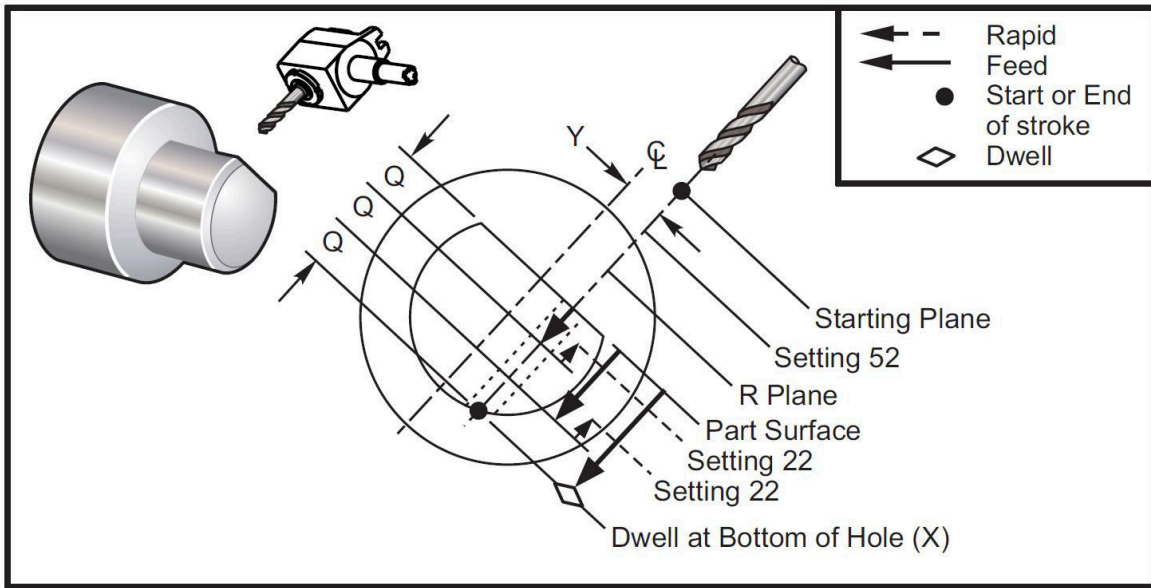
G243 Radial Normal Peck Drilling Canned Cycle

The following is a typical G243 Radial Normal Peck Drilling Canned Cycle:

```
(G243 - RADIAL PECK DRILLING USING Q)
G54                        (Work offset G54)
G00 G53 Y0                 (Home Y-axis)
G00 G53 X0
G00 G53 Z0
T303
M154                       (Engage C Axis)
M133 P2500 (2500 RPM)
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 Y0
G00 G53 X0
G00 G53 Z0
M00
```

For L11.01 machines and older, you can use standard axial canned cycles to drill radially, by changing the plane from G18 to G17, and changing your depths from a Z to an X diameter value.

The following radial drilling example is only to be used if you have version **L11.01 or older**.

```
%
O00831 (G83 - RADIAL PECK TEST)
(THIS PROGRAM DRILLS RADIALLY)
(IT IS ONLY VALID ON SOFTWARE)
(VERSIONS L11.01 AND OLDER)
(FOR L11.02 AND NEWER, USE G243)
G54
G53 G00 Y0.
G53 G00 X0.
G53 G00 Z-7.
T505 (DRILL)
G18
G98
M154                (Engage C-Axis)
G00 G54 X6. C0. Z1.
G00 X6. Z0.25
G97 P1500 M133
(M08)
(SWITCH TO G17 FOR RADIAL ON OLDER LATHE)
(THE DEPTH IS AN X VALUE, FOR RADIAL)
G17 G83 G98 X1. I1. J0.5 K0.5 R5. F10.
C135.
C225.
G00 G80 Z0.25
M155
M135
M09
G18 (RETURN TO G18)
G28 H0. (Unwind C-Axis)
G00 G53 Y0.
G00 G53 X0.
M30
%
```

This page intentionally left blank.

SECTION 6.   TOOL OFFSETS

SETTING Work Offsets WITH MISMATCHED TOOL T/H (T131/T101)

Dual Spindle DS machines will often have two or more tools sharing the same turret position.

T101

T131

Do not use tools with mismatched Tool and Offset numbers
(T131 vs. T101) to set a Work Offset (i.e., G54).

The O.D. turning tool facing the Main Spindle, in the picture above, is T101. This is Tool 1, using Offset 1.

The O.D. turning tool facing the Secondary Spindle is T131. This is Tool 1, using Offset 31. Any offset other than 1 could have been used.

The Haas lathe has 50 available Tool Offsets.

Currently (L11.10A) we are able to touch off tools whose T number matches its Offset number. In the case above, we could use the optional Automatic Tool Probe (ATP) for T101, but would touch off T131 manually (paper on part), for the Secondary Spindle operation.

When setting a Work Offset, the control uses the Tool Offset that corresponds with the current turret position. In our example, this is Tool Offset 1. If a tool is using an offset that does not match its turret position (like T131), it must not be used to set a Work Offset (Z FACE MEAS).

The Applications Department is currently reviewing test software that enhances the functionality of the Automatic Tool Presetter, and the way the control chooses which Tool Offset is used, when setting a Work Offset. Check the release notes in upcoming lathe software, to see if Software Issues Database Project #46118 is included.

Until that project is released, touch off all tools that face the secondary spindle manually – without the use of the tool presetter. If using the G55 Work Offset for the Secondary Spindle, leave the G55 Z Offset value at zero, and touch off each tool on the face of the part in the Secondary Spindle.
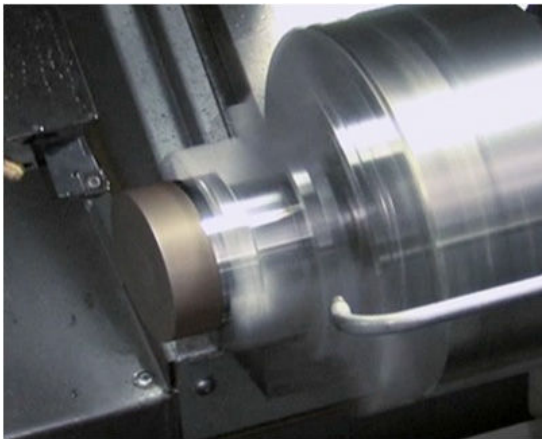
Touch off each of these tools on the face of the Secondary Spindle part. Do not use tools with mismatched tool/offset values to set a Work Offset.
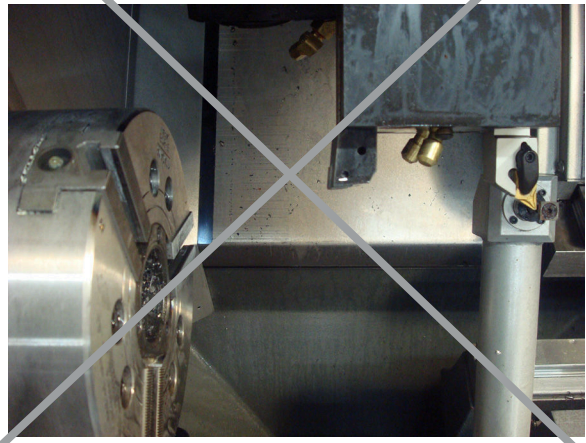
Do not use the Automatic Tool Presetter to set tools with mismatched tool/offset values. Touch off each of these tools on the face of the Secondary Spindle part.

SECTION 7.   AUTOMATIC TOOL PROBE (ATP)

See the 96-8700 Rev AN Lathe Manual, page 129, for information on the use of your Automatic Tool Setting Probe.

ES document ES0684 ST AUTO TOOL PROBE also contains very good information on the use of the tool probe.

Appendix D, at the end of this document, gives instructions for calibrating the ATP.

In a class setting, we will review the above documents making sure that everyone can:

1)   touch off tools manually, and from within a program

2)   program a broken tool detection cycle

3)   calibrate the ATP

4)   adjust probe Settings 59 and 60, "dialing in" the probe for accuracy.

If jogging manually into the probe, from HAND JOG, make sure you use the .001 or .0001 feed increment. Inaccurate probing can occur if handle-jogging into the probe at the .010 feed rate.

See the previous section for instructions on setting the Tool Offset of tools whose tool number and Tool Offset do not match (i.e., T131).

SECTION 8.   DUAL SPINDLE LATHES

COMMON DUAL SPINDLE (DS) M/G CODES

M10 – Main Spindle (CHUCK CLOSE)
M11 – Main Spindle (CHUCK OPEN)
M14 – Main Spindle (BRAKE ON)
M15 – Main Spindle (BRAKE OFF)
M03 (S) – Main Spindle Forward
M04 (S) – Main Spindle Reverse
M05 – Main Spindle Stop
M19 R0. – Main Spindle Orientation
(R0. = Orientation in Degrees)

M154 – C-axis Engage
M155 – C-axis Disengage
G199 – Engage Sync. Spindle Control
G198 – Disengage Sync. Spindle Control

M110 – Secondary Spindle (CHUCK CLOSE)
M111 – Secondary Spindle (CHUCK OPEN)
M114 – Secondary Spindle (BRAKE ON)
M115 – Secondary Spindle (BRAKE OFF)
M143 (P) – Secondary Spindle Forward
M144 (P) – Secondary Spindle Reverse
M145 – Secondary Spindle Stop
M119 R0. – Secondary Spindle Orientation
(R0. = Orientation in Degrees)

G04 P1. – Dwell (P1. = 1 second dwell)
G14 – Secondary Spindle Swap
G15 – Secondary Spindle Swap Cancel

M154 C-AXIS ENGAGE

To use the Main Spindle as a full 4th-axis rotary, we need to let the control know that we intend to use the Main Spindle as a C-Axis, and not as a lathe spindle (M03). We do this by commanding an M154 from the program or MDI.

Think of it this way:

M154 = Milling Mode. C-Axis rotary, like a 4th axis rotary on a mill.

M155 = Turning Mode. Standard spindle commands (M03, M05), like a lathe.

An M154 must be commanded from a program or MDI, to enable the C-Axis to be jogged. If the handwheel is rotated in HAND JOG Mode, with the C-Axis selected, and no spindle movement is seen, engage the C-Axis with an M154 command from MDI and continue.

The C-Axis is disengaged with an M155, or when the RESET button is pressed.

M154 C-Axis movements have an accuracy of +/-.01°, where M19 and M119 spindle orientations are accurate to within approximately +/-.15°.

The approximate diameter of the part, at the point where the tool makes contact with it, must be entered under Setting 102, C Axis Diameter. The control uses this value in its calculations, to maintain the proper G98 (Feed Per Minute) feedrate commanded by the programmer.

M14 / M114 SPINDLE BRAKE

Both the Main and the Secondary Spindles on a DS lathe make use of a mechanical brake.

Use the brake whenever live-tool milling or drilling is to be done, when simultaneous 4th-axis motion is not needed.

The Secondary Spindle can index with an M119 (G14 M19) command. It is not a full 4th-axis, and should be clamped whenever being used with live-tooling.

A dwell (typically 1.5 seconds, G04 P1.5) must be used, after an M119 orientation, to allow the motor time to settle into position, before the brake is applied, for accuracy.
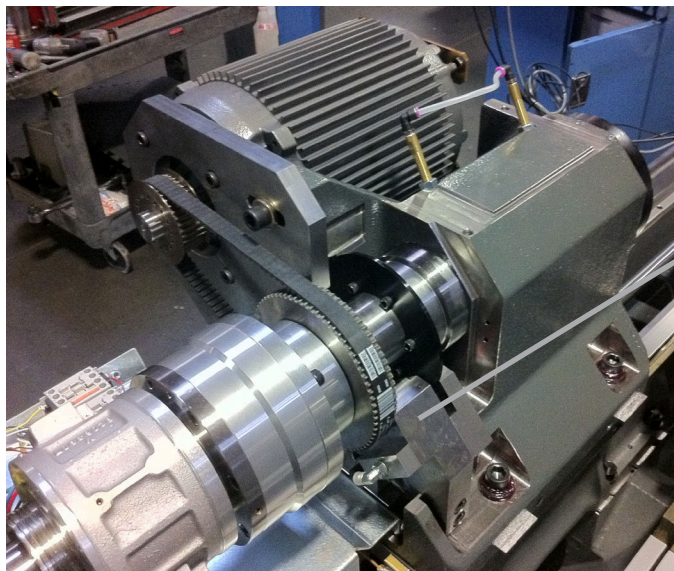
Secondary Spindle Brake

G04 DWELL

The G04 dwell code is used throughout Dual Spindle lathe programs.

When an M10/M110 Chuck Close command is given, it must be followed by a dwell to ensure that the part was fully clamped before machining begins.

When a part is handed-off from the Main to the Secondary Spindle, and an M11/M111 Chuck Open command is given, it must be followed by a dwell, giving the chuck enough time to fully release the part before a B-Axis movement is executed.

A dwell must be used, after an M119 spindle orientation, to allow the motor time to settle into position, before the brake is applied, for accuracy.

G04 P1.5 (DWELL FOR 1.5 SECONDS BEFORE CONTINUING)

G14, MACHINING ON SECONDARY SPINDLE

If doing work on the secondary spindle (canned cycles or turning), you would program the part to run on the main spindle, and then preface the code with a G14 (Secondary Spindle Swap).

Example of turning on the secondary spindle:

G14 (Secondary Spindle Swap)
G97 M03 S1000
G99 (Feed per rev.)
G01 Z-1. F.02
G15 (Cancel Secondary Spindle Swap)

All secondary spindle operations should contain a G14, followed by commands written for the main spindle (M03, G71 etc.), and ending with a G15 to cancel Secondary Spindle Swap.

If you get a 186 Spindle Not Turning alarm, it is likely because we are trying to cut on the Secondary Spindle without the use of a G14.

The G14 command tells the machine that we are using the secondary spindle. A G15 (Default, Secondary Spindle Swap Cancel), tells the machine we are using the main spindle.

Examples of turning on the main, turning on the secondary, and handing off a part, can be found in the lathe operator's manual in the Secondary Spindle Programming section. The manual can be downloaded from the Haas website.

G199 SPINDLE SYNCHRONIZATION

To get both spindles turning at the same time, we need just a couple of lines of code.

G97 M03 S1000 (START MAIN SPINDLE)
G199   (SYNC SPINDLES)

This is often used when cutting-off parts, held by both the Main and Secondary spindles. G199 can also be used for handing-off parts from one spindle to the next. To un-sync, the programmer will use a G198. Note that pressing Reset will not un-sync the spindles.

When transferring parts from the main to the secondary spindle, start the main spindle with a low rpm (G97 M03 S50), and then follow that with a Sync Command (G199). Both spindles will turn, with a matched rpm, and you can now bring in your B-Axis, and clamp/unclamp as necessary. It is not necessary to run the spindles when handing off parts.

When cutting off a part, you can start the main spindle (G97 M03 S1500), command Spindle Sync (G199), bring in your B-Axis with the secondary spindle chuck unclamped (M111), clamp on part, then command your cutoff tool as normal.

It may take up to 10 seconds for the spindles to reach their maximum synchronized speed of 3200 RPM, if starting from an RPM of 0, while synchronized within a G199. If both spindles are clamped on a part, they must ramp up-to-speed together, and this is unavoidable.

If the two spindles are not connected by a part, it will take less time to get to up-to-speed if you start each spindle independently, and then G199 Sync., like the following:

```
%
O00005
G198                (NO SYNC – BETTER NOT BE CLAMPED FROM BOTH ENDS)
G97 M03 S3000       (START MAIN)
G97 M144 P3000      (START SECONDARY SPINDLE)
G04 P1.
G199                (SYNC, IF YOU WANT, AFTER BOTH ARE AT FULL RPM)
G04 P1.
                    (RUN PART PROGRAM HERE)
G198                (NO SYNC – BETTER NOT BE CLAMPED FROM BOTH ENDS)
M05                 (STOP MAIN)
M145                (STOP SECONDARY SPINDLE)
M30
%
```

Getting up-to-speed from a dead stop is always faster when spindles are not synced.

R PHASE, SPINDLE SYNC (G199)

Open the Lathe Manual 96-8700AN now, and turn to the Dual-Spindle Lathes section. There is an excellent example of syncing the spindles on a DS.

Some parts require that the Main and Secondary Spindles mesh/orientate perfectly during a part handoff or machining operation. Two examples of this are:

1) Running custom shaped parts, like hexagon stock, that must feed precisely into a custom collet on the Secondary Spindle.

2) Running parts that are shorter than the height of your chuck jaws. The jaws from each chuck must mesh on these short parts.
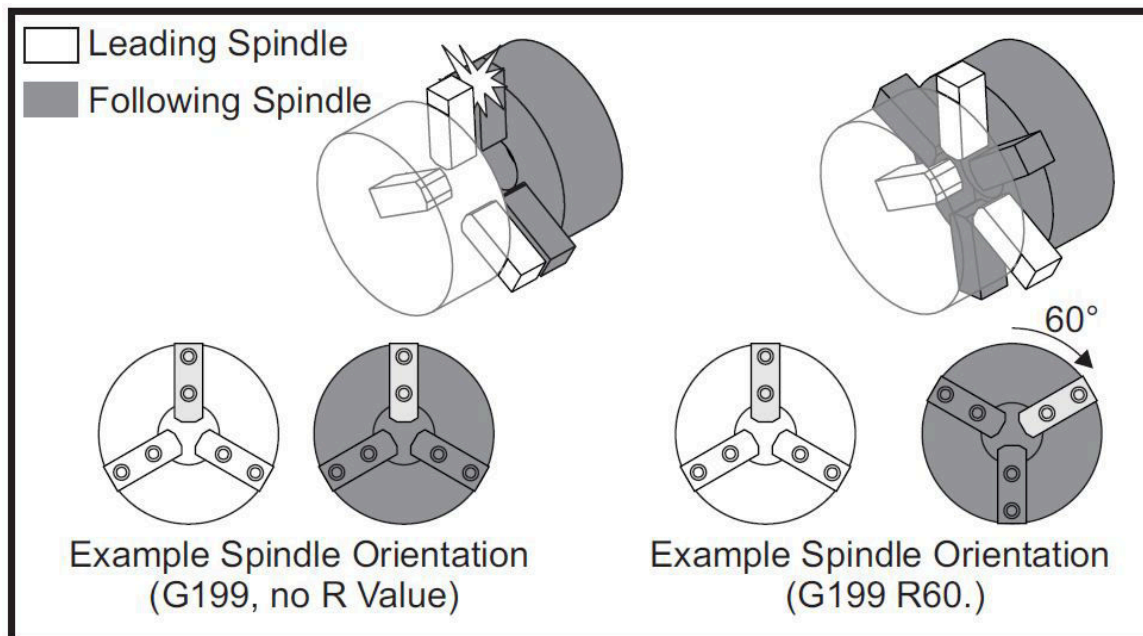
To find the proper alignment, we would:

1) Orientate both the main and secondary spindles with an M19 R0., and M119 R0.

2) Jog the B-Axis, bringing the two spindles near each other so the alignment can be viewed. Write down the current B-Axis Machine Position.

3) Engage the C-Axis (M154) and jog it until the main and secondary spindle chucks are aligned.

4) View and record the current C-Axis position, from the Position Screen (60.0 degrees for example).

5) This angle will be placed, as an R value, on your G199 line.

When you call up your G199 spindle sync, add that recorded angle to the G199 line as follows:

G199 R60.

A second method of determining what R Phase value you should use with your G199, is to:

1) Orientate both the main and secondary spindles with an M19 R0.; M10; M119 R0.
2) Jog the B-Axis, bringing the two spindles near each other so the alignment can be viewed.
3) Rotate the Secondary Spindle by hand, while slowly Handle-Jogging the B-axis until the chuck fully engages the part. Do this without moving the Main Spindle.
4) View and record the current SS position, from the SPINDLE SYNCHRONIZATION CONTROL Screen (60.0 degrees for example). This screen can be viewed by pressing CURNT COMDS and pressing Page Up until the Spindle Sync. page is seen.
5) This angle will be placed, as an R value, on your G199 line.

Test the alignment by calling up the following from MDI:

```
G97 M03 S50      (START MAIN SPINDLE)
G199 R60.        (SYNC SPINDLES WITH R PHASE)
G04 P10.         (DWELL TO SHOW ALIGNED SYNC)
G198            (SPINDLE SYNC OFF)
```

Every customer who buys a DS deserves to see this running on their new machine the first day. It is practical, and extremely cool looking. Practice this carefully before showing a customer.

Note that the spindles will remain synchronized, after a G199 call, until a G198 is called. Hitting the RESET button will not take the machine out of G199 Mode. To cancel, you must call up a G198 in your program, or in MDI.

Spindles can be synced without rotating. Calling a G199 R60. (from example above), without commanding the spindle to rotate (M03), will still sync the spindles.

When programming a DS lathe, it is easy to forget which way is positive for the C-Axis, Spindle Orientations, and Spindle Directions (M03 vs. M04). The chart below is a good resource for the programmer to have at his desk:

DS-30SSY

M3

| C0. OR M19 P0. | C90. OR M19 P90. | C180. M19 P180. | C270. OR M19 P270. |

MAIN SPINDLE

M143

| M119 P0. | M119 P90. | M119 P180. | M119 P270. |
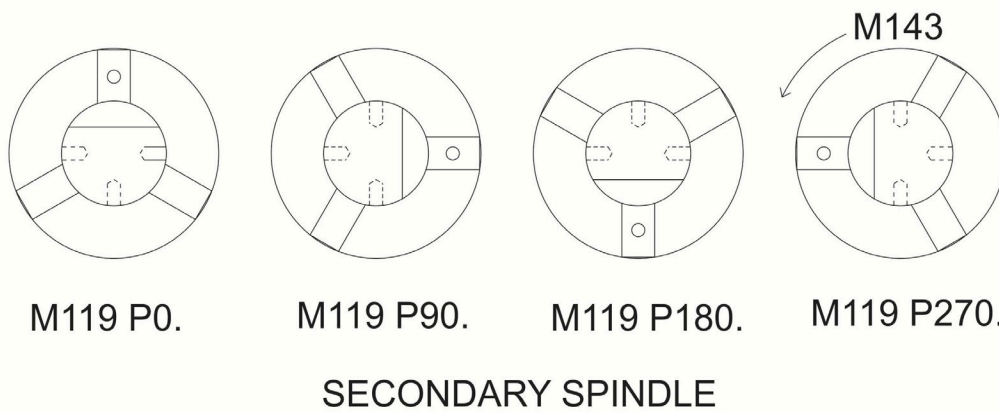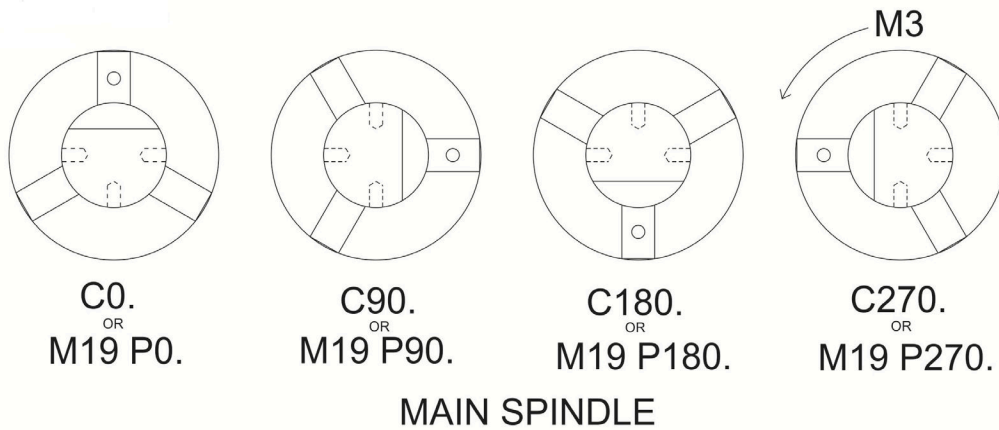
SECONDARY SPINDLE

SPINDLE OVERRIDES AND LOAD MONITORING

Secondary Spindle and Live Tool Loads can be seen by pressing the CURNT COMDS button, and paging up or down until the TOOL LOAD page is seen. These loads are displayed near the bottom of the upper-right pane.

Currently (L11.10A), Tool Load Monitoring is not available when machining on the Secondary Spindle, using a G14, or G199. When in G199, the spindles have a larger than normal load, which can cause an Overload Alarm, if this extra load was not anticipated.

There are currently no Spindle or Feed Overrides for the Secondary Spindle or live-tooling. There are open software projects which will add these features in a future release.

M19 VS M154 SPINDLE ORIENTATION

It is always best to use the M154 (C-Axis) to orientate the Main Spindle, as opposed to using an M19. This is a much more accurate and repeatable method. The Secondary Spindle on a DS machine has no C-Axis, and indexing on that spindle must be commanded with an M119, or a G14 M19.

M154 C-Axis movements have an accuracy of +/-.01°, where M19 and M119 spindle orientations are accurate within approximately +/-.100°.

The lathe manual lets us know that to position the secondary spindle, an M119 must be used. When orientating the Secondary Spindle to a known position, a "P" rounds to the nearest whole degree, and an "R" rounds to the nearest hundredth of a degree (x.xx). The angle is viewed in the Current Commands Tool Load screen.

If orientation is needed on the Secondary Spindle, the use of a dwell after the M119 will improve accuracy. This will give the spindle time to get into position before clamping.

```
G14           (SECONDARY SPIND SWAP ON)
M19 P0        (ORIENTATE SPINDLE)
G04 P1.5      (DWELL BEFORE CLAMP AFTER M19)
M14           (SPINDLE BRAKE ON)
              (ADD MACHINING OPERATION HERE)
G15           (SECONDARY SPIND SWAP OFF)
```

SECTION 9.	BAR FEEDER

INTRODUCTION

Download and read the Servo Bar 300 manual, 96-0013 Rev. BA, April 2012.

That document contains all of the information needed for Bar Feeder use, not covered in this short Bar Feeder Overview.

To view the V-axis from the Position Screen, we may need to change which (5) Axes are visible:

1) Press the POSIT key
2) Press the F2 key
3) Select up to 5 axes you would like to display, by highlighting them and then pressing the WRITE ENTER key
4) Press F2 when finished

Refer to Section 1 of this document, Homing the Machine, for more information on how to view the V-axis from the Position Screen.

COMMON BAR FEEDER CODES

G105

A G105 in a program will push the stock out by the amount in variable 3101 (#3101) the first time, and variable 3100 (#3100) each additional time it is called.

If variable 3110 (current bar length) is less than variable 3102 (Min Clamping Length), a G105 command will load a new bar and push it out to the value in variable 3101.

A G105 command is usually called-up near the end of the part program. If a bar has not been loaded yet for a job, the operator must command the initial G105 from MDI, or by beginning his part program from the G105 line in his program, this first time.

Q Mode Descriptions

These Q commands follow a G105 code, and can be used from within a program, or MDI.

Q0 Normal Bar Feed   **Q5** Set EOB Position
Q1 Set Bar Length   **Q6** Unload Push Rod
Q2 Set Reference Position   **Q7** Load Push Rod
(**Q2** Used In Combination with **Q4** Only)   **Q8** Unload Bar Stock
Q3 Set Alt Reference Position   **Q9** Load Bar Stock
Q4 Jog to Reference Position

Macro Variables

#3100 PART LENGTH + CUTOFF Bar feed increment (Length of bar pushed out each G105 after bar is loaded). Finished part length + cutoff length + face cleanup allowance.

#3101 INITIAL PUSH LENGTH Initial bar feed length (Length of a bar pushed out, past reference position, when loaded).

#3102 MIN CLAMPING LENGTH Minimum length for clamping (Length of bar required to support length pushed past the collet face).

#3103 MAX # PARTS Maximum number of parts.

#3104 MAX # BARS Maximum number of bars.

#3105 MAX LENGTH TO RUN Maximum length to run.

#3106 CURRENT # PARTS RUN Part counter.

#3107 CURRENT # BARS RUN Bar counter.

#3108 CURRENT LENGTH RUN Length counter.

#3109 LENGTH OF LONGEST BAR Length of the longest bar (set to 48 if unknown). Setting the length close to the size of the bar stock allows faster measurement of shorter bars. This length must be longer than the bar stock being used.

#3113 MIN RETRACT POSITION. Adjust this to make sure the push rod retracts out of the spindle liner after each G105 push. Jog the V-Axis until there is a safe gap between the end of the push rod and the spindle liner (approximately 1 inch/25 mm). Look at your V-Axis position, it will be a negative number (example: -13.0). Enter this number, as a positive value under #3113 (example: #3113=13.0).

Read-only:

#3110 CURRENT BAR LENGTH Current bar length measured by the machine.

Internal Only:

#3112 REFERENCE POSITION Established using G105 Q4 Jog to Reference Position, G105 Q2.

Entering Bar Feeder Values:

The bar feeder Macro Statements can be entered into the control via a Macro Statement, or directly onto the Bar Feeder 300 Page. To reach the Bar Feed page, press the CURNT CMND button and page up/down until the bar feeder variables are visible.

To change the values, highlight the variable (up/down cursor), enter a new value, and press WRITE/ENTER.

```
                        BAR FEEDER 300
HAAS SERVO BAR SYSTEM VARIABLES:

   3100    PART LENGTH + CUTOFF:          0.5100 in
   3101    INITIAL PUSH LENGTH:           1.2500 in
   3102    MIN CLAMPING LENGTH:           2.2500 in

   3103    MAX # PARTS:                         0
   3104    MAX # BARS:                          0
   3105    MAX LENGTH TO RUN:             0.0000 in

   3106    CURRENT # PARTS RUN:                 0
   3107    CURRENT # BARS RUN:                  1
   3108    CURRENT LENGTH RUN:            1.2500 in

   3109    LENGTH OF LONGEST BAR:        48.5000 in

   3110    CURRENT BAR LENGTH:           46.8290 in
   3113    MIN RETRACT POSITION:         13.0000 in
```

The following Macro Statement will enter a value of .510, into variable #3100, directly from a program or MDI:

#3100 = .510

Note that #3110 CURRENT BAR LENGTH is Read-Only, and can not be written to by this method.

BAR FEEDER APPLICATIONS QUICK-START GUIDE

Listed below, in **BOLD** print, are the common codes and variables that the user needs to command to use his bar feeder. Addressing each item, <u>in the order shown</u>, will cover most bar feeder applications.

A	Reference Position Macro Variable #3112
B	Initial Push Length Macro Variable #3101
C	Part Length + Cutoff Macro Variable #3100
D	Min Clamp Length Macro Variable #3102
E	Spindle Tool Clearance
F	Finished Part Length
G	Face Clean Up Allowance
H	Parting Tool Width
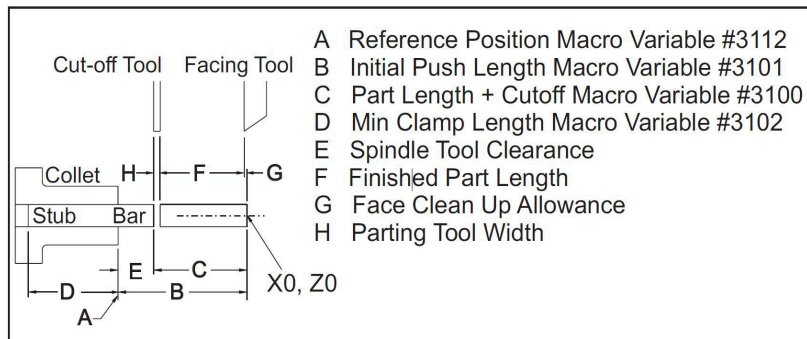
<u>**Sample Part Setup:**</u>

Part Length (F):	1"
Cutoff Tool Width (H):	.125"
Stock to Face (G):	.05"
Space Between Part and Chuck (E):	.5"
Cut Length of Bars:	44.0"

1) **G105 Q5** (calibrate bar feeder, done as needed and at initial machine setup)
   a. Load 12" bar for calibration
   b. Change Parameter 325 to match bar length, if not 12.0000"
   c. This only has to be done once in a while

2) **G105 Q4** (loads bar, jog stock to face of chuck/collet)
   a. Hit Reset
   b. Jog to face of chuck

3) **G105 Q2** (establishes the position of Collet/Chuck face. Only used with Q4)

4) **#3100** = 1.175 (part + cutoff+ amount to face) (1.+.125+.05)
   #3101 = 1.675 (#3100 + safe distance from chuck) (1.175+.5)
   #3102 = 2.00 (minimum amount to clamp on, usually .25" beyond collet clamping surface)
   #3109 = 45.0 (longest bar. Usually longest bar + 1.0") (44.0+1.0)
   #3113 = 11.5 (position of V-Axis, where pushrod is just clear of drawtube.
   Input as a positive number)

5) A **G105** in a program (or MDI) will push the stock by the value in #3101 the first time, and #3100 each additional time it is called

If #3110 (current bar length) is less than #3102 (Min Clamping Length), a G105 command will load a new bar, pushing it out past the collet face by the value in #3101.

Refer to Servo Bar 300 manual, 96-0013 Rev. BA, April 2012, for a full description of codes.

BAR PULLING WITH A DS MACHINE

Customers with Dual Spindle machines are able to "Pull" stock, by clamping on it with the Secondary Spindle, opening the Main Chuck, and commanding the B-Axis to the right.

When a bar is pulled by the Secondary Spindle on a DS lathe, the current bar length (variable #3110) is not updated. This movement is outside of the normal bar feeder operation.

A custom macro must be written, by the customer, for each application where the bar stock is re-positioned with the use of the secondary spindle chuck, or a bar-puller held in the turret.

When writing these programs, the programmer can call a G105 Q8 to clear the current bar. This will clear the current bar length (#3110), allowing for a new bar to be loaded the next time a G105 is called.

G160/G161 BAR FEEDER COMMANDS

Because the V-Axis is used as both our bar feeder Axis and for incremental Y-Axis movement, a special code is needed to let the control know which axis we intended to move. A G160 command, prior to a V move, lets the control know that the move is for the bar feeder, and not an incremental move of the Y-Axis. When the move is complete, a G161 must be called.

To move the bar feeder V-Axis 1" to the right of its Home Position (Absolute Move), we would command:

G160
G00 V-1.0
G161

Note that a G00 V-1.0 command, without the G160, would move the Y-Axis incrementally -1.0.

All V-axis Absolute Positions, right of the bar feeder's Home Switch, are commanded with negative values. For example V-10.0 would be 10.0" to the right of the home switch.
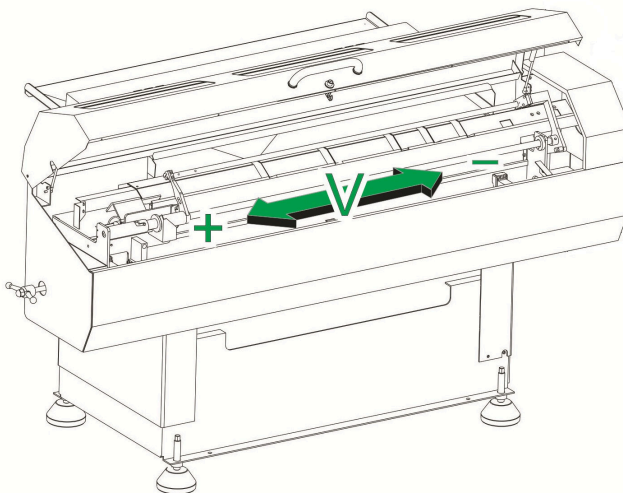
To move the bar feeder V-Axis 1" to the right of its present position (Incremental Move), we would command:

G105 J1.0

The Current Bar Length, #3110, is updated when the above G105 J1.0 command is called. Commanding the bar feeder V-Axis directly (G00 V-1.0), outside of the G105 command, will not update the Current Bar Length (#3110).

SECTION 10.   LIVE-TOOL MILLING, G17/G19

G17 (XY) PLANE AXIAL MILLING

In this section we are going to show three different methods of Axially Milling, in the G17 plane.

1) **Direct XY milling**. In this method we will program all arcs with X and Y movements.

2) **G112 XY to XC Interpretation**. Here, we take a mill program (written in G17 XY) and with the use of a G112, let the control convert it into XC movements.

3) **Direct XC milling**. This type of program makes simultaneous X and C-Axis moves.

DIRECT XY AXIAL MILLING, G17 (XY) PLANE

G17 is used when creating arcs (G02, G03) in the XY plane, for live-tool axial milling.

All X values are programmed in diameter. This applies to milling, as well as turning. The top of a 1.0" diameter bar in the chuck would be X1.0, the bottom of that bar would be X-1.0.

Incremental X-Axis moves are commanded with a U move. These are also in diameter.

If you command "G00 U-1.0" in MDI, the turret will move incrementally towards the spindle by .5", which is -1.0" diameter.

When milling in the G17 (XY) plane, and programming an arc with an "I" or an "R", we would use the actual size of the radius, not multiplying the "I" by 2, as if it were a diameter. This is the only X move that is not commanded as a diameter.
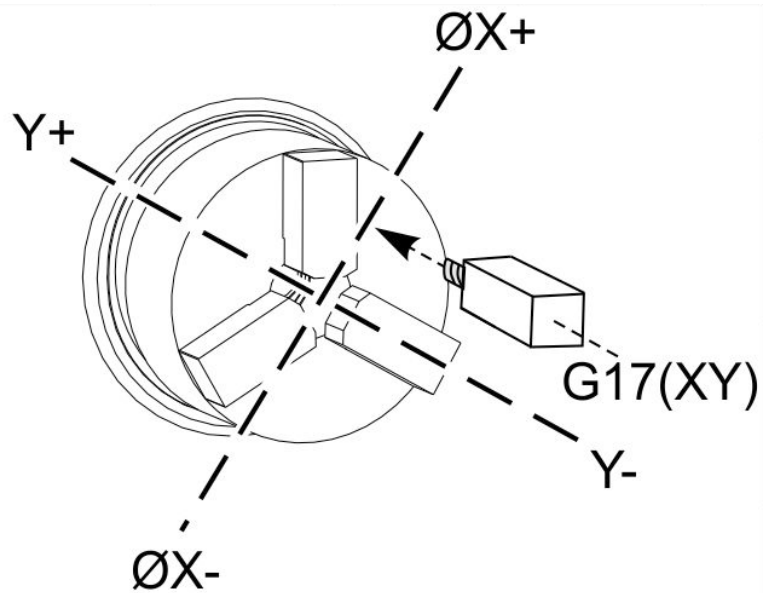
X-Axis Travel Limits

The program example on the following page must be run on an SS Y-Axis lathe, like an ST-20SSY. The extended X-Axis travel of an SS lathe allows the tool to reach further below centerline.

Most standard, non Super-Speed, Y-Axis lathes have less than the needed .625" of travel below X centerline, for the next mill example.

Larger parts can be easily programmed and run, when the programmer uses a G112 cycle. An example of G112 milling can be seen later in this section.

Using a G112 will be the most used method of XY G17 Axial Milling for the hand-programmer.

Each machine has different X-Axis travel limits. This is important when deciding what approach to take when live-tool milling. If you are axial-milling with a .5" diameter endmill, you will need to travel to X-1.5 in order to mill a 1" boss (remember, X values are in diameter). This may work well on an ST-20SSY, but a standard ST-20Y would not have the travel to reach that far below the X centerline.

In that instance we would program the boss using XC moves, or a G112 cycle.
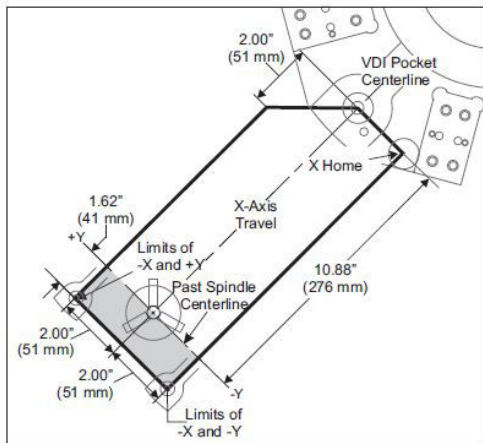
DS-30SSY Shown

Model Number	Travel Below X CL, Axial Live Tool	TS Morse Taper Size
OL-1	6.00	-
ST-10	0.35	3
ST-10Y	0.35	3
ST-20	0.40	4
ST-20Y	0.40	4
ST-20SS	1.70	4
ST-20SSY	1.70	4
ST-30	0.48	4
ST-30Y	0.48	4
ST-30SS	1.62	4
ST-30SSY	1.62	4
ST-40	1.20	5
ST-40L	1.20	5
DS-30	0.48	-
DS-30Y	0.48	-
DS-30SS	1.62	-
DS-30SSY	1.62	-

Values shown are actual inches below
X centerline, not diameter values
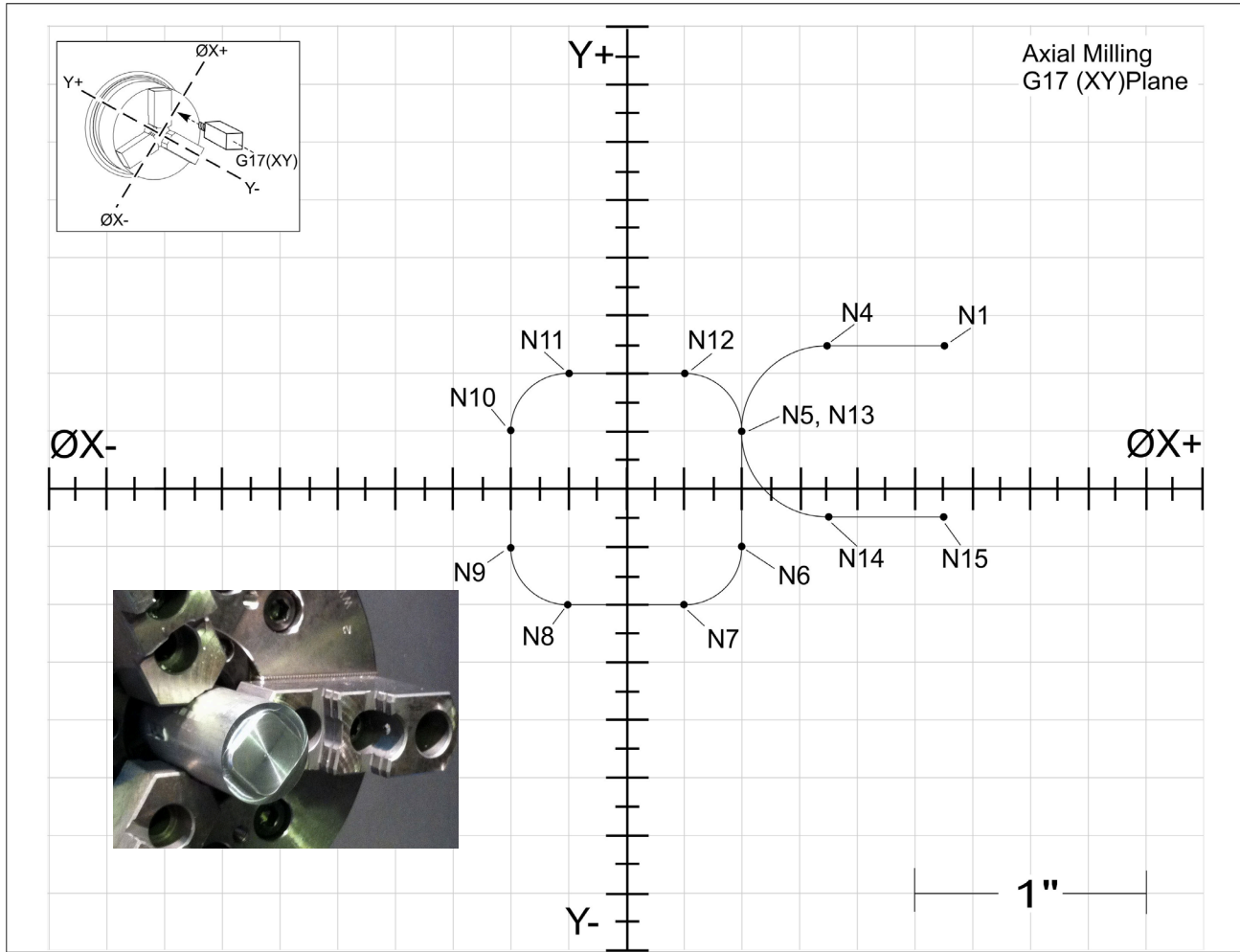
- **All X values are diameter values.**
- **All I, J, and R radius values are Incremental (not diameter)**
- **R values can be used instead of I, J values.**
- **We chose to approach the part from the X+ direction with our tool. Approaching from the X- or Y+/- direction is more likely to result in an over-travel issue.**
- **All Haas Y-axis lathes have +/- 2.0" of travel in the Y-Axis.**

```
N1 X2.75 Y0.625
N2 Z0.1 (CLEARANCE PLANE)
N3 G01 Z-0.05 F10. (Z FINAL DEPTH)
N4 G41 X1.75 F20. (LEADIN WITH COMP)
N5 G03 X1. Y0.25 R.375 (I0 J-0.375)
N6 G01 Y-0.25
N7 G02 X0.5 Y-0.5 R.25 (I-0.25 J0)
N8 G01 X-0.5
N9 G02 X-1. Y-0.25 R.25 (I0 J0.25)
N10 G01 Y0.25
N11 G02 X-0.5 Y0.5 R.25 (I0.25 J0)
N12 G01 X0.5
N13 G02 X1. Y0.25 R.25 (I0 J-0.25)
N14 G03 X1.75 Y-0.125 R.375(I0.375 J0)
N15 G40 G01 X2.75 (LEADOUT)
```

See full program on next page

```
%
O00017 (G17 AXIAL MILL TEST)

(CUTS 1IN SQUARE)
(WITH .25IN CORNER RAD)
(1.25 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)
(.5IN EM)
T101
M01
G97 M133 P3000
G98 (IN PER MIN)

N1 X2.75 Y0.625
N2 Z0.1 (CLEARANCE PLANE)
N3 G01 Z-0.05 F10. (Z FINAL DEPTH)
N4 G41 X1.75 F20. (LEADIN WITH COMP)
N5 G03 X1. Y0.25 R.375 (I0 J-0.375)
N6 G01 Y-0.25
N7 G02 X0.5 Y-0.5 R.25 (I-0.25 J0)
N8 G01 X-0.5
N9 G02 X-1. Y-0.25 R.25 (I0 J0.25)
N10 G01 Y0.25
N11 G02 X-0.5 Y0.5 R.25 (I0.25 J0)
N12 G01 X0.5
N13 G02 X1. Y0.25 R.25 (I0 J-0.25)
N14 G03 X1.75 Y-0.125 R.375 (I0.375 J0)
N15 G40 G01 X2.75 (LEADOUT)

G00 Z1.
G53 G00 Y0.
G53 G00 X0.
M135
M01
G18 (RETURN TO G18 PLANE)
M30
%
```
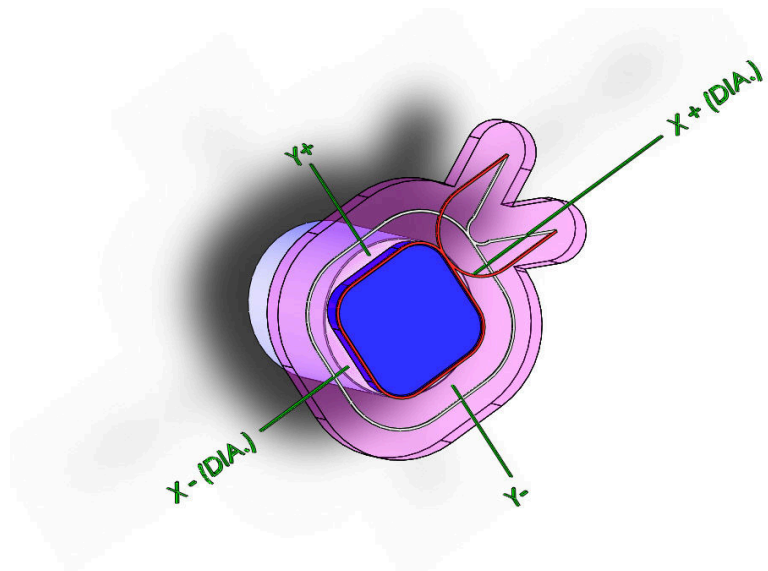
See Appendix C for more G17 milling examples.

See Appendix G for printable plane-specific (G17/G19) Graph Paper.

G112 XY TO XC INTERPRETATION

For this example we will program the same 1" square, with .25" corner radius, for a Vertical Mill, and then use that code to create a G112 XC program for our lathe.

No Y-Axis is needed for this cycle. It can be performed on Haas lathes with or without a Y-Axis, using only the X, C, and Z axes
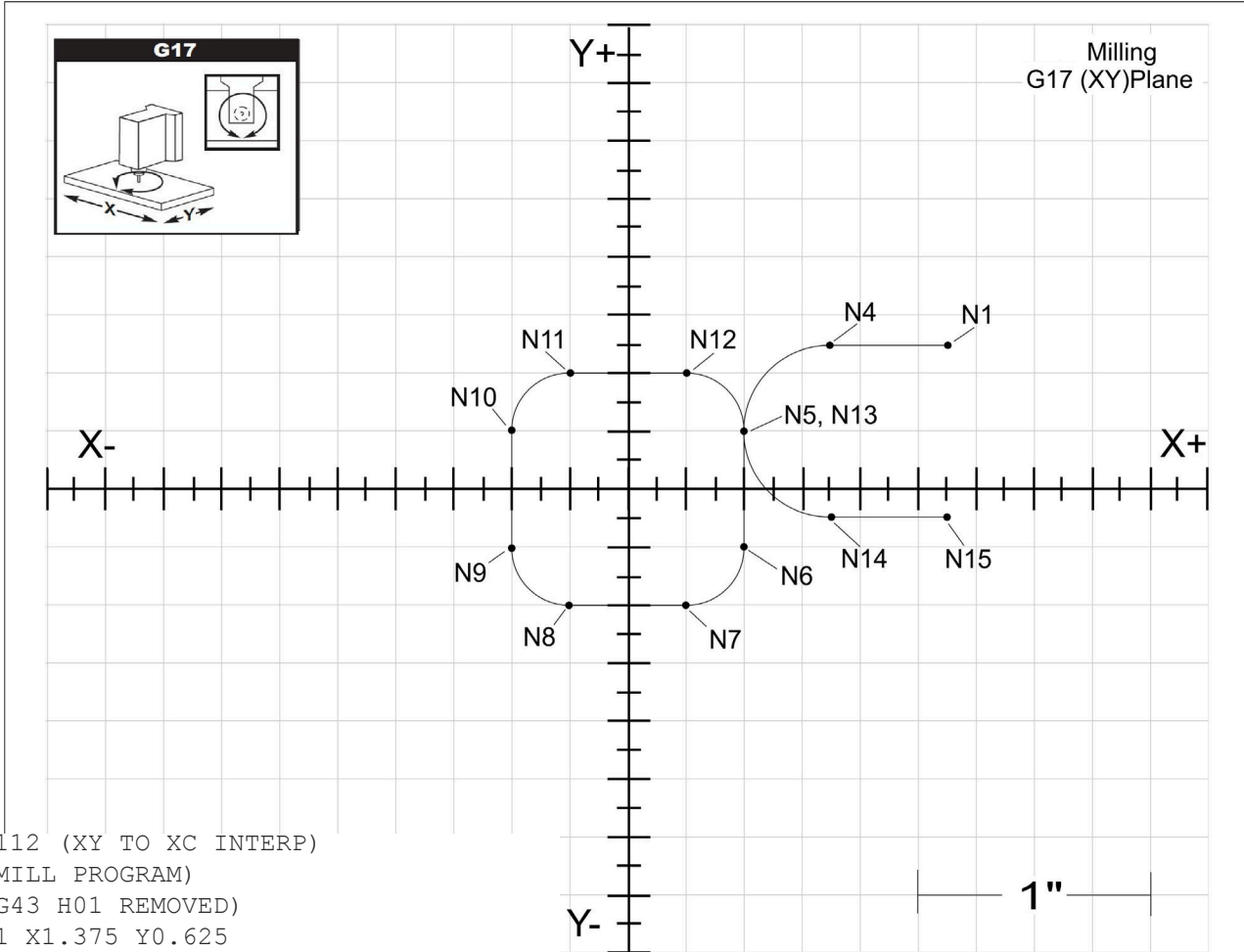

```
G112 (XY TO XC INTERP)
(MILL PROGRAM)
(G43 H01 REMOVED)
N1 X1.375 Y0.625
N2 Z.5 (Z CLEARANCE PLANE)
N3 G01 Z-.05 F10. (Z FINAL DEPTH)
N4 G41 X0.875 F20. (LEADIN WITH COMP)
N5 G03 X0.5 Y0.25 R0.375
N6 G01 Y-0.25
N7 G02 X0.25 Y-0.5 R0.25
N8 G01 X-0.25
N9 G02 X-0.5 Y-0.25 R0.25
N10 G01 Y0.25
N11 G02 X-0.25 Y0.5 R0.25
N12 G01 X0.25
N13 G02 X0.5 Y0.25 R0.25
N14 G03 X0.875 Y-0.125 R0.375
N15 G40 G01 X1.375 (LEADOUT)
G113 (CANCEL G112)
```

See full program on next page

- **The M06 T1 from the mill program was replaced with a T101, for a lathe.**
- **The M03 spindle command was replaced with an M133 P spindle command for live-tooling.**
- **The G43 and H01 were removed from the original mill program.**
- **We chose to approach the part from the X+ direction with our tool. Approaching from the X- or Y+/- direction is more likely to result in an over-travel issue.**
- **Having the tool enter the part from the X- direction, with a G112, could cause the toolpath to violate the contour of the part.**
- **Only the X-Axis will move, while the C-Axis rotates to cut your part.**

See Appendix C for more G17 milling examples.

See Appendix G for printable plane-specific (G17/G19) Graph Paper.

```
%
O00112 (G112 RADIAL MILL TEST)

(CUTS 1IN SQUARE)
(WITH .25IN CORNER RAD)

(T1 = .5IN ENDMILL)
(SET TOOL TO .25 RADIUS)
(ON TOOL OFFSET PAGE)

G53 G00 Y0.
G53 G00 X0.
G53 G00 Z0.

G00 G54

(.5IN EM)
T101
M154 (ENGAGE C-AXIS)
M01
G97 M133 P2000
G98 (IN PER MIN)

G17 (SELECTS G17 XY PLANE)
G112 (XY TO XC INTERP)
(MILL PROGRAM)
(G43 H01 REMOVED)
N1 X1.375 Y0.625
N2 Z.5 (Z CLEARANCE PLANE)
N3 G01 Z-.05 F10. (Z FINAL DEPTH)
N4 G41 X0.875 F20. (LEADIN WITH COMP)
N5 G03 X0.5 Y0.25 R0.375
N6 G01 Y-0.25
N7 G02 X0.25 Y-0.5 R0.25
N8 G01 X-0.25
N9 G02 X-0.5 Y-0.25 R0.25
N10 G01 Y0.25
N11 G02 X-0.25 Y0.5 R0.25
N12 G01 X0.25
N13 G02 X0.5 Y0.25 R0.25
N14 G03 X0.875 Y-0.125 R0.375
N15 G40 G01 X1.375 (LEADOUT)
G113 (CANCEL G112)

G00 Z1. (Z CLEARANCE PLANE)
G53 G00 Y0.
G53 G00 X0.
M135
M01
G18 (RETURN TO G18 PLANE)
M30
%
```

DIRECT XC AXIAL MILLING, G17 (XY) PLANE

Example

```
%
O0054
T101
G54
M133 P2000
M154 (Engage C-axis)
G00 G98 (feed/min) X2.0 Z0
C90
G01 Z-0.1 F6.0     (position 1)
X1.0               (position 2)
C180. F10.0        (position 3)
X2.0               (position 1)
G00 Z0.5
M155
M135
G53 X0
G53 Z0
M30
%
```
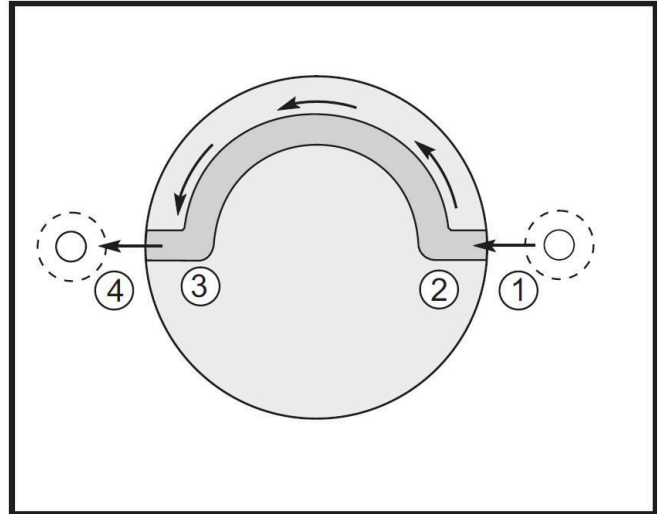
ALIGNMENT OF AXIAL CANNED CYCLES, G17 MILLING

Axial Canned Cycles often need to be aligned with G17 Axially-Milled features.

C0.0 (M154; G00 C0.0) aligns with the X+ axis, as shown below.

This alignment is maintained by the machine whether the G17 milling was programmed directly with XY moves, or run with a G112 XY-to-XC cycle.
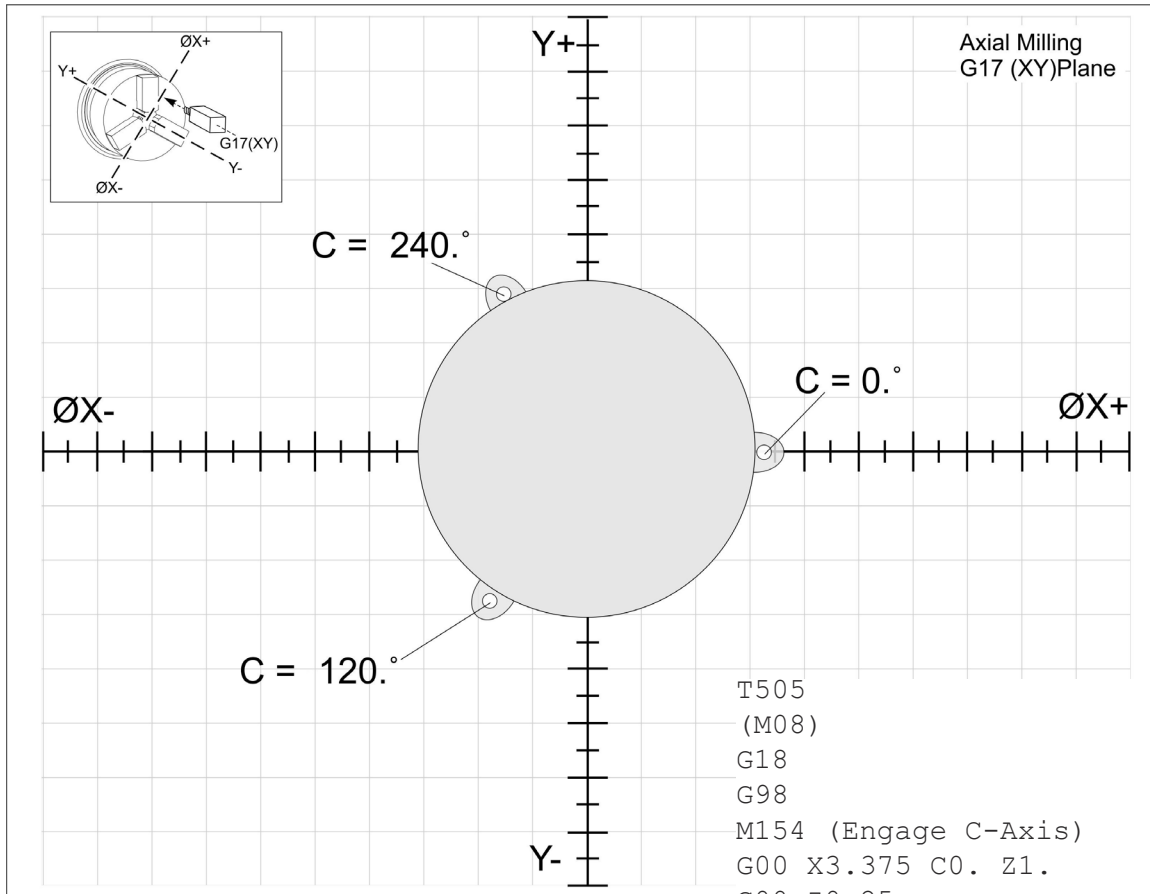
The full G112 part program, pictured above, can be found in Appendix C of this document.

```
T505
(M08)
G18
G98
M154 (Engage C-Axis)
G00 X3.375 C0. Z1.
G00 Z0.25
G97 P1500 M133
G83 G98 X3.375 C0. Q0.1
R0.1 Z-0.4 F10.
C120.
C240.
G00 G80 Z0.25
M155
M135
```

G19 (YZ) RADIAL LIVE-TOOLING MILLING

The following program example will mill a 1" square, with .25" corner radius, radially, in the G19 (YZ) plane.
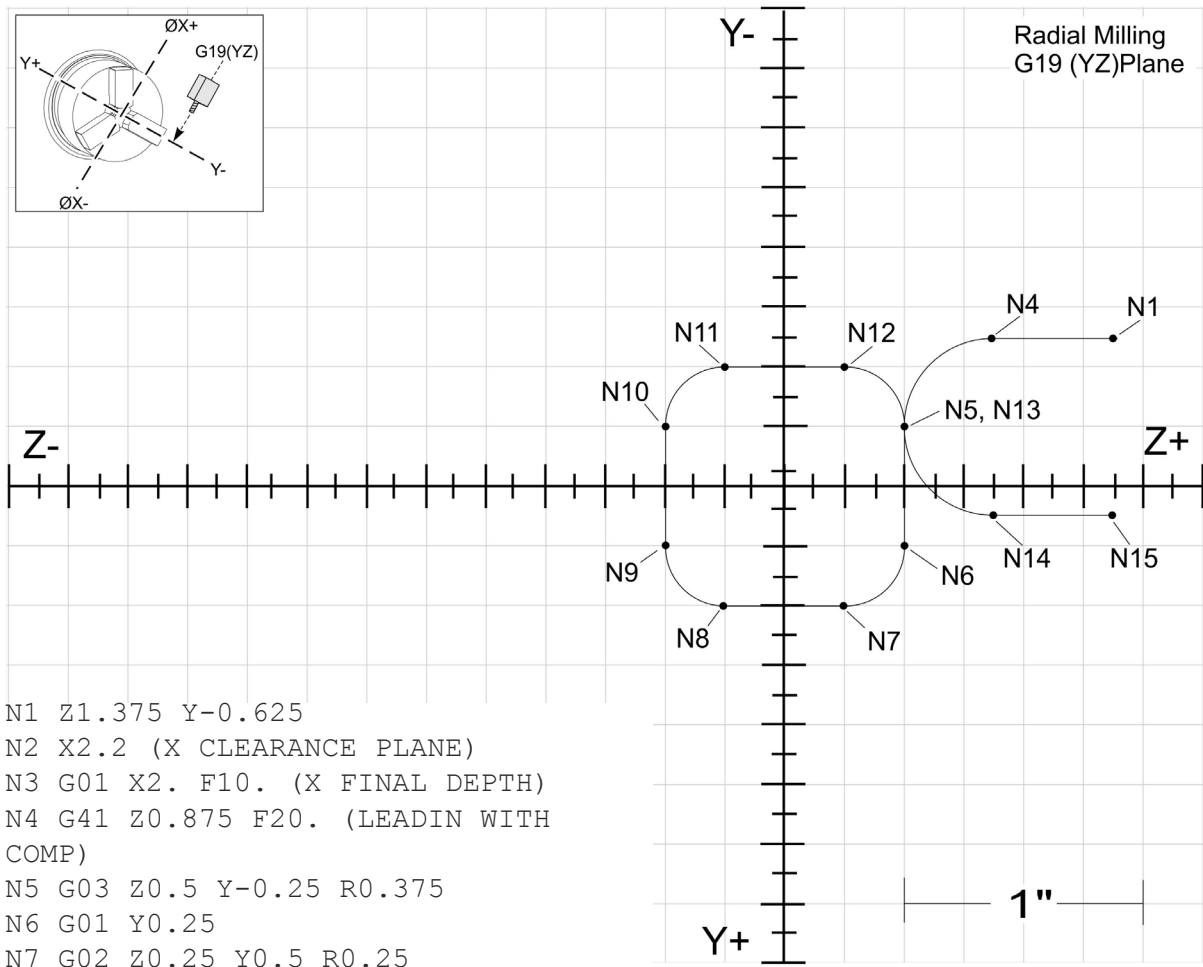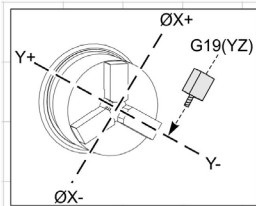

```
N1 Z1.375 Y-0.625
N2 X2.2 (X CLEARANCE PLANE)
N3 G01 X2. F10. (X FINAL DEPTH)
N4 G41 Z0.875 F20. (LEADIN WITH
COMP)
N5 G03 Z0.5 Y-0.25 R0.375
N6 G01 Y0.25
N7 G02 Z0.25 Y0.5 R0.25
N8 G01 Z-0.25
N9 G02 Z-0.5 Y0.25 R0.25
N10 G01 Y-0.25
N11 G02 Z-0.25 Y-0.5 R0.25
N12 G01 Z0.25
N13 G02 Z0.5 Y-0.25 R0.25
N14 G03 Z0.875 Y0.125 R0.375
N15 G40 G01 Z1.375 (LEADOUT)
```

See full program on next page

- **Note that Y+ is towards the operator, unlike a vertical mill.**
- **All J, K, and R radius values are Incremental (not diameter).**
- **R values can be used instead of J, K values.**
- **All Haas Y-axis lathes have +/- 2.0" of travel in the Y-axis.**

```
%
O00019 (G19 RADIAL MILL TEST)

(CUTS 1IN SQUARE)
(WITH .25IN CORNER RAD)

(T1 = .5IN ENDMILL)
(SET TOOL TO .25 RADIUS)
(ON TOOL OFFSET PAGE)

G53 G00 Y0.
G53 G00 X0.

G00 G54
M154
M10
G19 (SELECTS G19 YZ PLANE)
(.5IN EM)
T101
M01
G97 M133 P2000
G98 (IN PER MIN)

N1 Z1.375 Y-0.625
N2 X2.2 (X CLEARANCE PLANE)
N3 G01 X2. F10. (X FINAL DEPTH)
N4 G41 Z0.875 F20. (LEADIN WITH COMP)
N5 G03 Z0.5 Y-0.25 R0.375
N6 G01 Y0.25
N7 G02 Z0.25 Y0.5 R0.25
N8 G01 Z-0.25
N9 G02 Z-0.5 Y0.25 R0.25
N10 G01 Y-0.25
N11 G02 Z-0.25 Y-0.5 R0.25
N12 G01 Z0.25
N13 G02 Z0.5 Y-0.25 R0.25
N14 G03 Z0.875 Y0.125 R0.375
N15 G40 G01 Z1.375 (LEADOUT)

G00 X3.
G53 G00 Y0.
G53 G00 X0.
M135
M155
M01
G18 (RETURN TO G18 PLANE)
M30
%
```
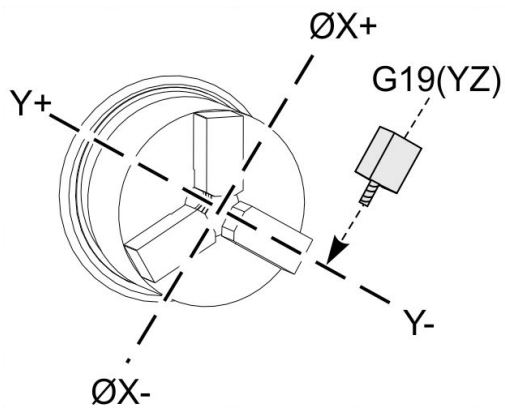
SECTION 11.   M38 / M39 SSV SPINDLE SPEED VARIATION

Varying the spindle RPM, while turning a part, can decrease chatter.

The RPM changes help avoid frequencies where the part and machine would naturally resonate.

The Haas Control has a built in feature which will adjust your RPM, during the cycle, to avoid this vibration.

If you are running a long, or chatter prone part, try the following settings, with an M38 code.

M38 SSV

Par. 166 = 10
(1 second)

Par. 165 = 200
(+/- 200 RPM)

RPM
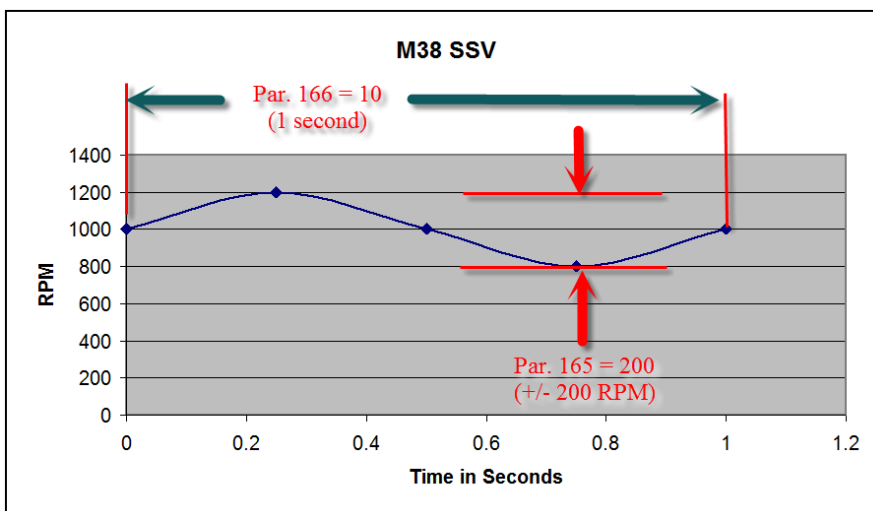
Time in Seconds

Setting 165 SSV VARIATION = 200. This will adjust your RPM by 200 revolutions above and below your programmed speed.

Setting 166 SSV CYCLE (0.1) SECS = 10. This defines how long it takes, in seconds, for the RPM changes (Set. 165) to take place. Values entered are in 1/10th seconds, where a 10 equals one second.

Try different values, finding a frequency where the chatter does not occur. Your chipload, G99 Feed/Rev., will be maintained by the control.

Example:

```
O0010;
S1000 M3
G04 P3.
M38 (SSV ON)
G04 P60.
M39 (SSV OFF)
M30
```

The above part was machined, using the M38 SSV feature

This page intentionally left blank.

SECTION 12.	SETTING XY CENTERLINE

INSTRUCTIONS FOR SETTING TURRET POCKET X AND Y AXIS CENTERLINES, VDI/BOT/HYBRID

1) Load a VDI Axial ER holder, or a BOT boring bar holder into the turret. If using a BOT holder, make sure the Y-adjustment cams are in a neutral position. See picture below. If there are two cam adjustments, make sure they are even with each other, and in their neutral positions (not high or low). If you have a Hybrid Turret, use a VDI holder.
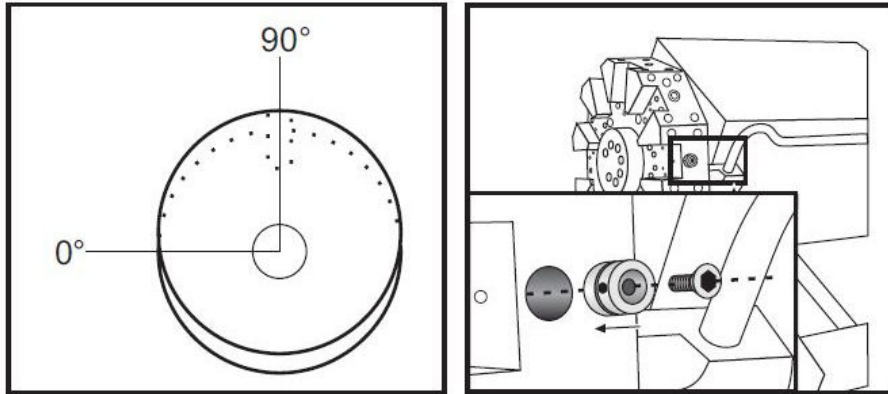
2) Set Parameter 211 Y TOOL CHANGE OFFSET to 0, restart machine.

3) Load a Co-Axial indicator into the Main Spindle Chuck/Collet. If you don't own a Co-Ax indicator, buy one. Your results will likely be off by about .001" if using a standard indicator with a mag-base (indicator droop).

4) Sweep the ground I.D. of your BOT/VDI tool holder with your indicator, adjusting the X and Y axis until you have found the center of the holder. Note that you can not indicate in a blank VDI pocket, those holes are eccentric by design. You must indicate in on the ground I.D. of a holder, like an axial VDI ER-32.

5) Enter DEBUG Mode (LIST PROG, ALARM MESGS, enter "DEBUG", WRITE/ENTER).

6) Go to the POSIT screen. The XY position will be listed in the upper left-hand corner of the screen.

7) We will set our Y value first. Our example Debug Raw Data Y Actual was 3.8175.

   a. Remove the decimal place, leaving us with 38175.

   b. Multiply this by 1.4142. This is because the Y axis is a compound axis. We would get 53987.085. We only need the whole number, which leaves us with 53987.

   c. Invert the sign, leaving us with -53987, and enter this value into Parameter 211 Y TOOL CHANGE OFFSET.

```
 COMMON  SOFTWARE  INPUT INVERT   SYSTEM    AXIS  COMPENSATION

 Y  PARAM  B

   114 Y   ACC/DEC T CONST                           200
   122 Y   PHASE OFFSET                                0
   126 Y   GRID OFFSET                             -86869
   135 Y   EXACT STOP DIST.                          180
   139 Y   FRICTION COMPENSAT.                         0
   146 Y   ACCEL FEED FORWARD                          0
   132 Y   SCREW COMP. COEF.                           0
   203 Y   AIR BRAKE DELAY                             0
   211 Y   TOOL CHANGE OFFSET                      -53979
   230 Y   LEAD COMP 10E9                              0
   273 Y   SCREW COMP T. CONST                         0
   280 Y   SCALE GAIN MULT.                            0
   286 Y   LINEAR SCREW OFFS                           0
   328 Y   SCALES PER INCH                             0
   334 Y   SCALES PER REV                              0
   340 Y   SPINDLE THERM COEF.                         0
  1065 Y   FEEDBACK NOTCH FILTER DEPTH                 0
  1081 Y   FEEDBACK NOTCH FILTER FREQUENCY             0
  1097 Y   FEEDBACK NOTCH FILTER ROLL OFF              0




 Parameter 211 Y TOOL CHANGE OFFSET




   X   Y   Z   A   B   Sp   U   V   W   C   TT   SS
```

8) Restart the machine, and repeat steps 4-6 from above, again.

9) We will now set our X axis centerline. This will be the value entered as your X Tool Geometry, when the F2 button is pressed.

10) Our example Debug Raw Data X Actual value (see picture on step 6) was -905.6854.

   a. Remove the decimal place, leaving us with -9056854.

   b. Invert the sign, leaving us with 9056854, and enter this value into Parameter 254 SPINDLE CENTER.

```
COMMON  SOFTWARE  INPUT INVERT   SYSTEM   AXIS  COMPENSATION

SPINDLE

  131      MAX SPINDLE RPM                          4800
  150      MAX SP RPM LOW GEAR                      1073
   73      SP HIGH G/MIN SPEED                        48
   74      SP LOW G/MIN SPEED                         80
   79      SPINDLE STEPS/REV                        8192
  239      SPNDL ENC STEPS/REV                      8192
  119      SPINDLE DECEL DELAY                        35
  120      SPINDLE ACC/DECEL                         25
  304      SPINDLE BRAKE DLY                        500
  254      SPINDLE CENTER                       9056852
  238      SPINDLE AT SPEED %                       100
  600      PEAK SPIN. PWR - KW                       30
   70      MIN SPIN DELAY TIME                       50
  118      SPINDLE REV TIME                          30
  838      SP ORIENT LP CUTOFF                        0
  840      SP ORIENT I GAIN                           0
  842      SP P GAIN SCL FACTR                        0
  844      SP DYN GAIN TYPE                           0
  725      SP HIGH SPEED CURRENT PROFILING            0



Parameter 254 SPINDLE CENTER
```

11) Restart the machine.

   a. Call up the holder you were indicating (T101), in MDI.

   b. In HAND JOG Mode, press OFFSET, and with your tool (T101) highlighted, press F2. This will write the X centerline value of your holder to the X Geometry column for your tool.

TOOL 7	TOOL GEOMETRY					
OFFSET	X	Y	Z	RADIUS	TAPER	TIP
1	-5.0000	0.	0.	0.	0.	0
2	-15.3089	0.	-0.0002	0.	0.	3
3	-5.4000	0.	-0.0003	0.	0.	0
4	-5.4000	0.	-0.0004	0.	0.	0
5	-5.4000	0.	-0.0005	0.	0.	0
6	-5.4000	0.	-0.0006	0.	0.	0
7 ACTIVE	-21.7612	0.	-0.0007	0.	0.	0
8	0.	0.	-0.0008	0.	0.	0
9	0.	0.	-0.0009	0.	0.	0
10	-20.0000	0.	-0.0010	0.	0.	0

ENTER A VALUE - F2 FOR X OFFSET TO CENTER LINE

   c. Enter a value of 0, to the Y Geometry column, for your tool.

   d. Make sure there is no X or Y value in your G54 Work Offset.

   e. From MDI, command: T101; G00 G54 X0 Y0. The tool will move into position.

12) Indicate in the holder once again, to verify that it is in position.

13) If it is off by a small amount, you can add or subtract encoder counts to your Parameter 211 (Y) and 254 (X) as needed.

   a. You can view your position in debug again, and add/subtract the encoder counts you see. For Y, you would need to multiply the Y ACTUAL you see by 1.4142 (no decimal places), and then add/subtract that value to Parameter 211.

   b. You can repeat step 10 from above again to get closer, or just add/subject encoder counts to Parameter 254 until your holder indicates-in true.

FREQUENTLY ASKED QUESTIONS

Q: When I try to use a Live-Tool Canned Cycle, the main spindle turns instead of my live-tooling.

A: Check Parameter 315 Bit 1 NO SPIND CAN CYCLE. This needs to be set to a "1" for all Live-Tooling lathes. Make sure you are using the correct canned cycle. See the Live-Tool Canned Cycle sections of this document. Make sure you are using an M133 Pxxxx and not an M03 Sxxxx.

Q: I get a "CHUCK OPEN" message displayed, and the machine stops moving, when pressing Cycle Start when the chuck is open. This can happen when I press Feed Hold in a barfeed operation, when trying to barpull, or when single-blocking through a program (chuck open).

A: Check Parameter 278 Bit 8 CHUCK OPEN CSTART. This should be on. Restart the machine after changing this parameter. In lathe versions L11.02-L11.05 this function was limited, as a safety measure. The solution is to either: 1) Turn the SETUP KEY to the on position to continue, or 2) upgrade your software to L11.10A or newer where this feature has been restored.

Q: We are getting a 186 SPINDLE NOT TURNING Alarm, on our DS lathe when trying to turn on the second spindle.

A: All machining done on the secondary spindle requires the programmer to use a G14 Secondary Spindle Swap code. Program the second spindle like you would the main spindle, and then preface the code with a G14, and end it with a G15.

Q: When I tap or drill with a Live-Tool Canned cycle, it moves in the wrong direction, axially instead of radially, or radially instead of axially.

A: Check your plane selection and your choice of canned cycles. We have cycles that run either Axially or Radially. Since L11.06 your plane selection is disregarded for canned cycles (G17, G18), so we need to make sure we use the correct cycle.

If you are running software older than L11.06, then your plane selection is very important, for those machines we use a G18 to drill/tap axially and a G17 to drill/tap radially. Plane selection is still critical for live-tool milling operations, no matter what software version you are running.

Axial Cycles: G81, G82, G83, G85, G86, G87, G88, G89, G95, and G186

Radial Canned Cycles: G195, G196, G241, G242, G243, G245, and G249

Q: When I machine a circle with axial live-tooling (G02/G03), it comes out more like a triangle. What could it be?

A: Take a look at the Live-Tool Milling section of this document, and see if your program looks correct. If the code looks good, the control could be changing your plane before intended. There was a look-ahead issue in early L11.xx series software (prior to L11.06), where the control could change planes too soon, when another plane was called out later in your program. The solution is to either: 1) stop look-ahead before your plane change by placing an M01, /, M00, or G103 P1;;; before any plane change (G17, G18, G19) or 2) Upgrade your software to the latest L11.10A software or newer.

Q: What is the largest tap size I should use, when using live-tooling?

A: We consistently tap 5/8-11 holes with live tooling (on mild steel). Once you go to a 3/4" thread and larger, you should consider threadmilling.

Q: We are getting a 186 SPINDLE NOT TURNING Alarm, when trying to tap using Live-Tooling.

A: This can happen if the Rigid Tapping Option (Parameter 57 Bit 4) has been turned off. If the Parameters on a machine are loaded/restored manually (not during the DEBUG>UPDATE process), all paid Option Codes will be turned off.

Q: Do you have proofed canned cycle programs for Live-Tooling lathes?

A: Yes. See appendix A and B at the end of this document.

Q: Can we contact Haas directly for Applications help?

A: Absolutely. We can be reached by email at applications@haascnc.com, or give us a call at 805-278-8500.

FUTURE ENHANCEMENTS

The following are open projects. These enhancements are in the system, requested by customers and dealers, and are being worked on for future release. They are not currently available in L11.10A.

- Feed and Spindle Speed Overrides for Live-Tooling and Second Spindle (DS)
- Work Offset setting with tools that have mismatched Tool/Offset (T130 vs. T101)
- The display of more than 5 axes on the position screen at once
- Restore POWER UP RESTART function on DS lathes, as opposed to ZERO RET>SINGL
- The use of a G28 V0.0 to return the Y-Axis to the home position will be available. Currently we would use a G53 G00 Y0.

APPENDIX A.   AXIAL LIVE-TOOL CANNED CYCLES

G81 DRILL AXIAL

```
%
O00810 (G81 AXIAL LIVE-TOOL)
 (AXIAL FACE DRILL TEST)

G53 G00 Y0.
G53 G00 X0.

T2323
G18
G98
M154 (Engage C-Axis)
G00 G58 X6. C0. Y0. Z1.
G00 X2. Z0.25
G97 P1500 M133
 (M08)
G18 G81 G98 X1. Z-1. F10.
C135. Y-1.
C225. Y1.
C315.
G00 G80 Z0.25
M155
M135
M09
G18
G28 H0. (Unwind C-Axis)
G00 G58 X6. Y0 Z1.
G18 (Return to XZ plane)
G99 M01
M30
%
```

G82 SPOT DRILL AXIAL

```
%
O00820 (G82 AXIAL LIVE-TOOL)
 (G82 SPOT DRILL CYCLE)

G00 G53 Y0.
G00 G53 X0.

 (AXIAL FACE DRILL TEST)
T2323
G18
G98
M154 (Engage C-Axis)
G00 G58 X6. C0. Y0. Z1.
G00 X1.5 Z1.
G97 P1500 M133
 (M08)
G82 G98 R0.25 X2. Z-1. F10. P1000
C135. X2.5 Y-1.
C225. X1. Y1.
C315.
G00 G80 Z0.25
M155
M135
M09
G18
G28 H0. (Unwind C-Axis)
G00 G58 X6. Y0 Z1.
G18 (Return to XZ plane)
G99 M01
M30
%
```

G83 PECK DRILL AXIAL

```
%
O000830 (G83 AXIAL LIVE-TOOL)

G58
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.

T2323
G18
G98
M154 (Engage C-Axis)
G00 G58 X6. C0. Y0. Z1.
G00 X1.5 Z0.25
G97 P1500 M133
(M08)
G18 G83 G98 C45. Z-1. R0.25 F10.
Q0.125
C135. Y-1.
C225. Y1.
C315.
G00 G80 Z0.25
M155
M135
M09
G28 H0. (Unwind C-Axis)
G00 G53 Y0.
G00 G53 X0.
G99
M30
%
```

G85 BORING AXIAL

```
%
O000850 (G85 AXIAL LIVE-TOOL)
(G85 BORING CANNED CYCLE)

G00 G53 Y0.
G00 G53 X0.

T2323
G18
G98
M154 (Engage C-Axis)
G00 G58 X6. C0. Y0. Z1.
G00 X1.5 Z1.
G97 P1500 M133
(M08)
(WATCH YOUR PLANE G17 G19 BAD)
G18 (G17) G85 G98 R0.25 X2. Z-1. F10.
U-1.
C135. X2.5 Y-1.
C225. X1. Y1.
C315.
G00 G80 Z0.25
M155
M135
M09
G18
G28 H0. (Unwind C-Axis)
G00 G58 X6. Y0 Z1.
G18 (Return to XZ plane)
G99 M01
M30
%
```

G86 BORE AND STOP AXIAL

```
%
O000860 (G86 AXIAL LIVE-TOOL)
(G86 BORE AND STOP CANNED CYCLE)

G00 G53 Y0.
G00 G53 X0.

T2323
G18
G98
M154 (Engage C-Axis)
G00 G58 X6. C0. Y0. Z1.
G00 X1.5 Z1.
G97 P1500 M133
(M08)
G18 G86 G98 R0.25 X2. Z-1. F10.
U-1.
C135. X2.5 Y-1.
C225. X1. Y1.
C315.
G00 G80 Z0.25
M155
M135
M09
G18
G28 H0. (Unwind C-Axis)
G00 G58 X6. Y0 Z1.
G18 (Return to XZ plane)
G99 M01
M30
%
```

G87 BORE, MANUAL RETRACT AXIAL

```
%
O000870 (G87 AXIAL LIVE-TOOL)
(G87 BORE AND MANUAL RETRACT)

G00 G53 Y0.
G00 G53 X0.

T2323
G18
G98
M154 (Engage C-Axis)
G00 G58 X6. C0. Y0. Z1.
G00 X1.5 Z1.
G97 P1500 M133
(M08)
G18 G87 G98 R0.25 X2. Z-1. F10.
U-1.
C135. X2.5 Y-1.
C225. X1. Y1.
C315.
G00 G80 Z0.25
M155
M135
M09
G18
G28 H0. (Unwind C-Axis)
G00 G58 X6. Y0 Z1.
G18 (Return to XZ plane)
G99 M01
M30
%
```

G88 BORE, DWELL, MANUAL RETRACT AXIAL

```
%
O000880 (G88 AXIAL LIVE-TOOL)
(G88 BORE DWELL MAN. RETRACT)

G00 G53 Y0.
G00 G53 X0.

(AXIAL FACE DRILL TEST)
T2323
G18
G98
M154 (Engage C-Axis)
G00 G58 X6. C0. Y0. Z1.
G00 X1.5 Z1.
G97 P1500 M133
(M08)
G18 G88 G98 R0.25 X2. Z-1. F10.
P2000
U-1.
C135. X2.5 Y-1.
C225. X1. Y1.
C315.
G00 G80 Z0.25
M155
M135
M09
G18
G28 H0. (Unwind C-Axis)
G00 G58 X6. Y0 Z1.
G18 (Return to XZ plane)
G99 M01
M30
%
```

G89 BORE AND DWELL AXIAL

```
%
O000890 (G89 AXIAL LIVE-TOOL)
(G89 BORE AND DWELL CYCLE)

G00 G53 Y0.
G00 G53 X0.

T2323
G18
G98
M154 (Engage C-Axis)
G00 G58 X6. C0. Y0. Z1.
G00 X1.5 Z1.
G97 P1500 M133
(M08)
G18 G89 G98 R0.25 X2. Z-1. F10. P2000
U-1.
C135. X2.5 Y-1.
C225. X1. Y1.
C315.
G00 G80 Z0.25
M155
M135
M09
G18
G28 H0. (Unwind C-Axis)
G00 G58 X6. Y0 Z1.
G18 (Return to XZ plane)
G99 M01
M30
%
```

G95 FORWARD RIGID TAP RH THREAD AXIAL

```
%
O000950 (G95 AXIAL LIVE-TOOL)
(G95 AXIAL LIVE-TOOL TAP)
(1/4 x 20 Tap)

G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.

T2323
G18
G99
M154 (ENGAGE C-AXIS)
G00 G58 X6. C0. Y0. Z1.
G00 X1.5 Z0.5
(M8)
S500
G95 C45. Z-0.5 R0.5 F0.05
C135.
Y1.
Y-1.
G00 G80 Z0.5 M09
M135
M155
G28 H0. (Unwind C-Axis)
G00 G53 Y0.
G00 G53 X0.
G18 (Return to XZ plane)
M30
%
```

G186 REVERSE RIGID TAP, LH THREAD AXIAL

```
%
O01860 (G186 AXIAL LIVE-TOOL)
(G186 REVERSE AXIAL LIVE-TOOL TAP)

(LEFT HAND FACE TAP)

G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.

T2323
G18
G99
M154 (ENGAGE C-AXIS)
G00 G58 X6. C0. Y0. Z1.
G00 X1.5 Z0.5
(M8)
S500
G186 C45. Z-0.5 R0.5 F0.05
C135.
Y1.
Y-1.
G00 G80 Z0.5 M09
M135
M155
G28 H0. (Unwind C-Axis)
G00 G53 Y0.
G00 G53 X0.
G18 (Return to XZ plane)
M30
%
```

Haas Technical Publications

This page intentionally left blank.

APPENDIX B.   RADIAL LIVE-TOOL CANNED CYCLES

Radial Live-Tool Canned Cycles: G195, G196, G241, G242, G243, G245, and G249

G195 FORWARD LIVE-TOOL RADIAL TAPPING

```
%
O01950 ( G195 - RADIAL RH TAPPING )
G00 G58
G98
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.
T505
M154
M133 P500
G00 G53 X-2.
X6. Z0. Y-2. C0. (APPROACH
POSITION. START OF TAP CYCLE)
G99 (CALLS UP FEED PER REV. NEEDED
FOR G195)
S800 (NEEDED FOR TAPPING. SAME FOR
G195/G196)
G195 X3. F0.05 (TAPPING LINE. THIS
CALLS THE END POINT OF TAP)
C90. Y0. (APPROACH POSITION. START
OF TAP CYCLE)
G195 X3. F0.05
G00 G80
C0. (UNWIND C-AXIS)
G00 G53 Y0.
G00 G53 X0.
M135
M30
%
```

G196 REVERSE LIVE-TOOL RADIAL TAPPING

```
%
O01960 ( G196 - RADIAL LH TAPPING )
G00 G58
G98
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.
T505
M154
M133 P500
G00 G53 X-2.
X6. Z0. Y-2. C0. (APPROACH POSITION.
START OF TAP CYCLE)
G99 (CALLS UP FEED PER REV. NEEDED FOR
G195)
S800 (NEEDED FOR TAPPING. SAME FOR
G195/G196)
G196 X3. F0.05 (TAPPING LINE. THIS
CALLS THE END POINT OF TAP)
C90. Y0. (APPROACH POSITION. START OF
TAP CYCLE)
G195 X3. F0.05
G00 G80
C0. (UNWIND C-AXIS)
G00 G53 Y0.
G00 G53 X0.
M135
M30
%
```

G241 RADIAL DRILL

```
%
O02410( G241 - RADIAL DRILLING)
G54
G00 G53 Y0
G00 G53 X0 Z-7.
T303
M19 P0
M133 P500
G98
G00 X6. Z-0.75 Y-1.
G241 R4. X2. F20.
Y1. Z-1.2
G00 G80 Z1.
G00 G53 X0. Y0.
M135
M30
%
```

G242 RADIAL SPOT DRILL

```
%
O02420 ( G242 - RADIAL SPOT DRILL
)
G00 G58
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.
T505
M154
M133 P500
G98
G00 G53 X-2.
G00 C0.
G01 X6. Z0. F30.
G00
G242 R4. X3. P500 F30.
C180.
Y-2.
G00 G80
G00 C0.
G00 G53 X0. Y0.
M135
M30
%
```

G243 RADIAL NORMAL PECK DRILLING

```
%
O02430 ( G243 - RADIAL PECK
DRILLING )
G00 G58
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.
T505
M154
M133 P500
G98
G00 G53 X-2.
G00 C0.
G01 X6. Z0. Y0. C0. F30.
G00
G243 R5. X3. F30. I0.5 J0.2 K0.2
Y-2.
Y0. Z-1. C180.
G00 G80
G00 C0.
G00 G53 X0. Y0.
M135
M30
%
```

G245 RADIAL BORING

```
%
O02450 ( G245 - RADIAL BORING )
G00 G58
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.
T505
M154
M133 P500
G98
G00 G53 X-2.
G01 X6. Z0. Y0. C0. F30.
G00
G245 R4. X2. F30.
Y-1. Z-1. C180.
G00 G80
G00 C0.
G00 G53 X0. Y0.
M135
M30
%
```

G246 RADIAL BORE AND STOP

```
%
O02460 ( G246 - RADIAL BORE AND STOP
)
G00 G58
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.
T505
M154
M133 P500
G98
G00 G53 X-2.
G01 X6. Z0. Y0. C0. F30.
G00
G246 R4. X3. F30.
Z-1. Y-1. C180.
G00 G80
G00 C0.
G00 G53 X0. Y0.
M135
M30
%
```

G247 RADIAL BORE AND MANUAL RETRACT

```
%
O02470 ( G247 - RADIAL BORE AND MAN
RETRACT )
G00 G58
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.
T505
M154
M133 P500
G98
G00 G53 X-2.
G00 C0.
G01 X6. Z0. Y0. C0. F30.
G00
G247 R4. X3. F30.
Y-2. Z-1. C180.
G00 G80
G00 C0.
G00 G53 X0. Y0.
M135
M30
%
```

G248 RADIAL BORE, DWELL AND MANUAL RETRACT

```
%
O02480 ( G248 - RADIAL BORE DWELL &
MAN RETR )
G00 G58
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.
T505
M154
M133 P500
G98
G00 G53 X-2.
G00 C0.
G01 X6. Z0. Y0. C0. F30.
G00
G248 R4. X3. F30. P1000
Z-1. Y-1. C180.
G00 G80
G00 C0.
G00 G53 X0. Y0.
M135
M30
%
```

G249 RADIAL BORE AND DWELL

```
%
O02490 ( G249 - RADIAL BORE AND
DWELL )
G00 G58
G00 G53 Y0.
G00 G53 X0.
G00 G53 Z-7.
T505
M154
M133 P500
G98
G00 G53 X-2.
G00 C0.
G01 X6. Z0. Y0. C0. F30.
G00
G249 R4. X3. F30. P1000 L3
G00 G80
G00 C0.
G00 G53 X0. Y0.
M135
M30
%
```

This page intentionally left blank.
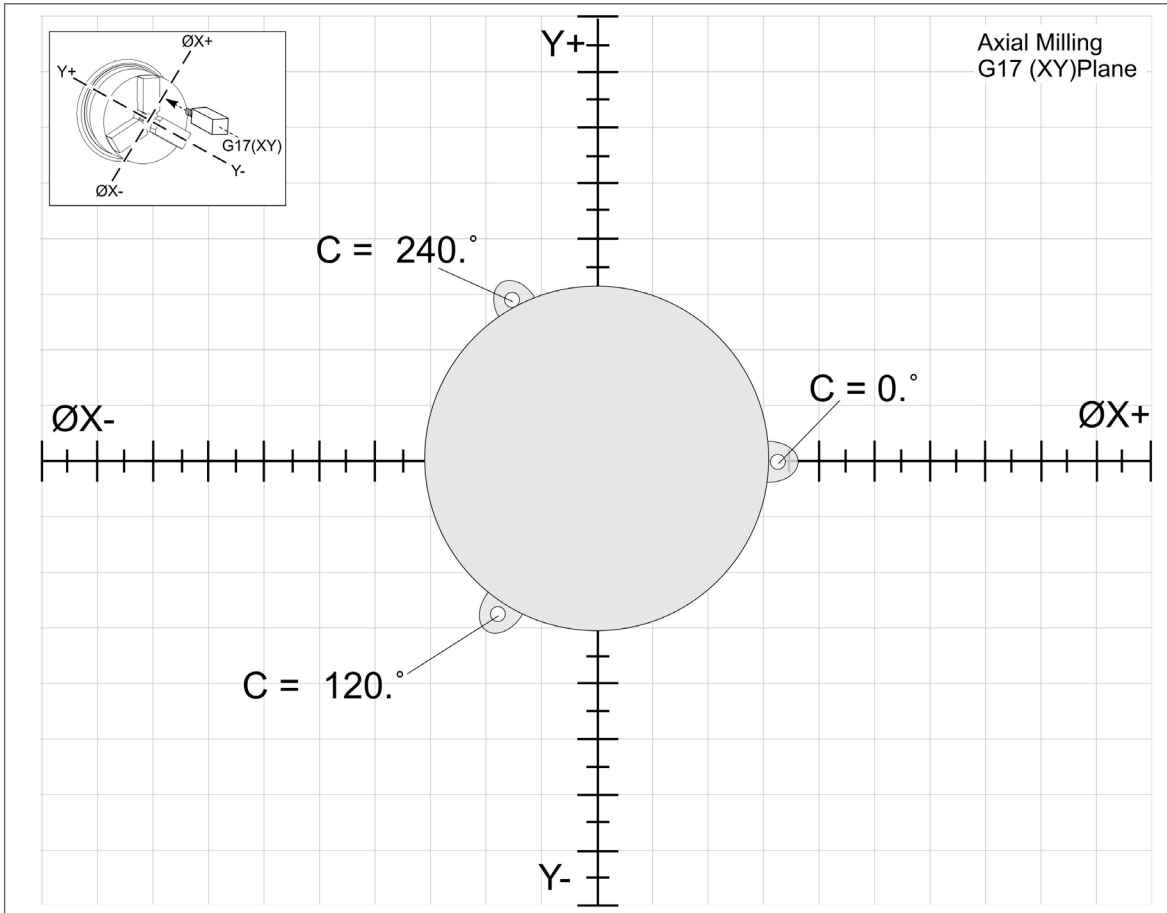
APPENDIX C.   G17 LIVE-TOOL MILLING EXAMPLE

G112 PROGRAM, WITH AXIAL CANNED CYCLE

The following G112 program was based on a mill program, and adapted to run on a live-tooling lathe.

The live-tool canned cycle drilling, aligns with the G112 Axial Milling, as shown in the graphic above.

Original Mill Code:

```
%
O0001 (MILL CODE TO CONTOUR)
(THIS CODE IS FOR A MILL)

M00
(DONT USE THIS CODE)
(THIS IS THE MILL CODE)
(THE G112 CYCLE IS BASED ON)

G00 G90 G40 G80 G54
(.5 IN EM)
T01 M06
M01
S3000 M03
(SEMI-FIN PASS)
(.01 IN XY ALLOWANCE)
X2.61 Y.1
G43 Z.1 H01 M08
G01 Z-.28 F20.
G41 X2.235 Y.1D01 F40.
G03 X2.135 Y0 I0 J-.1
G02 X1.8721 Y-.4076 I-.4475 J0
G03 X1.8637 Y-.418 I.0062 J-.0137
G02 X-.5698 Y-1.823 I-1.8637 J.418
G03 X-.5831 Y-1.8251 I-.0045 J-.0143
G02 X-1.2891 Y-1.4175 I-.2607 J.3637
G03 X-1.2939 Y-1.405 I-.0149 J.0015
G02 Y1.405 I1.2939 J1.405
G03 X-1.2891 Y1.4175 I-.0102 J.011
G02 X-.5831 Y1.8251 I.4453 J.0439
G03 X-.5698 Y1.823 I.0087 J.0122
G02 X1.8637 Y.418 I.5698 J-1.823
G03 X1.8721 Y.4076 I.0146 J.0033
```

```
G02 X2.135 Y0 I-.1846 J-.4076
G03 X2.235 Y-.1 I.1 J0
G40 G01 X2.61
G00 Z.1

(FINISH PASS)
(0 XY ALLOWANCE)
X2.6 Y.1
G01 Z-.28 F20.
G41 X2.225 Y.1D01 F40.
G03 X2.125 Y0 I0 J-.1
G02 X1.868 Y-.3985 I-.4375 J0
G03 X1.8539 Y-.4158 I.0103 J-.0228
G02 X-.5669 Y-1.8135 I-1.8539 J.4158
G03 X-.5889 Y-1.817 I-.0075 J-.0239
G02 X-1.2791 Y-1.4185 I-.2549 J.3556
G03 X-1.2871 Y-1.3976 I-.0249 J.0025
G02 Y1.3976 I1.2871 J1.3976
G03 X-1.2791 Y1.4185 I-.0169 J.0184
G02 X-.5889 Y1.817 I.4354 J.0429
G03 X-.5669 Y1.8135 I.0146 J.0203
G02 X1.8539 Y.4158 I.5669 J-1.8135
G03 X1.868 Y.3985 I.0244 J.0055
G02 X2.125 Y0 I-.1805 J-.3985
G03 X2.225 Y-.1 I.1 J0
G40 G01 X2.6
G00 Z.1 M09
G91 G28 Z0
M05
M30
%
```
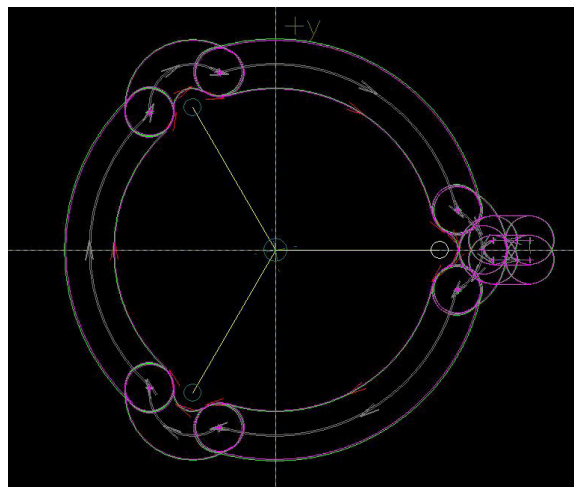
Final G112 Lathe Code, with Axial Canned Cycle:

```
%
O01778 (G112 AXIAL MILL TEST )
(DRILLS, CUTS 3 EARED BOSS)

(***** TOOL LIST ****)
(T1111 = .5 IN EM)
(T505 = .1875 IN DRILL)
(****** END LIST ****)

(****** DRILLING *******)
G54 G00
G53 G00 X0.
G53 G00 Z-7.
T505
(M08)
G18
G98
M154 (Engage C-Axis)
G00 X3.375 C0. Z1.
G00 Z0.25
G97 P1500 M133
G83 G98 X3.375 C0. Q0.1 R0.1 Z-0.4
F10.
C120.
C240.
G00 G80 Z0.25
M155
M135
M09
G28 H0. (Unwind C-Axis)

G53 G00 X0.
G53 G00 Z-7.


(******* AXIAL MILLING G112 *****)

G54 G00
(.5IN EM)
T1111
M154
(M08)
M01
G97 M133 P3000
G98 (IN PER MIN)
(SEMI-FIN PASS )
(.01 IN XY ALLOWANCE)

Z1.
X6. (POSITION TOOL)
G112 (XY TO XC INTERP)
G17 (SET TO XY PLANE)

X2.61 Y0.1
(G43) Z0.1 (H01) (REMOVED G43 FROM
MILL CODE)
G01 Z-0.28 F20.
G41 X2.235 Y0.1 D1 F40.
G03 X2.135 Y0 I0 J-0.1
G02 X1.8721 Y-0.4076 I-0.4475 J0
G03 X1.8637 Y-0.418 I0.0062 J-0.0137
G02 X-0.5698 Y-1.823 I-1.8637 J0.418
G03 X-0.5831 Y-1.8251 I-0.0045
J-0.0143
G02 X-1.2891 Y-1.4175 I-0.2607 J0.3637
G03 X-1.2939 Y-1.405 I-0.0149 J0.0015
G02 Y1.405 I1.2939 J1.405
G03 X-1.2891 Y1.4175 I-0.0102 J0.011
G02 X-0.5831 Y1.8251 I0.4453 J0.0439
G03 X-0.5698 Y1.823 I0.0087 J0.0122
G02 X1.8637 Y0.418 I0.5698 J-1.823
G03 X1.8721 Y0.4076 I0.0146 J0.0033
G02 X2.135 Y0 I-0.1846 J-0.4076
G03 X2.235 Y-0.1 I0.1 J0
G40 G01 X2.61
G00 Z0.1
(FINISH PASS)
(0 XY ALLOWANCE)
X2.6 Y0.1
G01 Z-0.28 F20.
G41 X2.225 Y0.1 D1 F40.
G03 X2.125 Y0 I0 J-0.1
G02 X1.868 Y-0.3985 I-0.4375 J0
G03 X1.8539 Y-0.4158 I0.0103 J-0.0228
G02 X-0.5669 Y-1.8135 I-1.8539 J0.4158
G03 X-0.5889 Y-1.817 I-0.0075 J-0.0239
G02 X-1.2791 Y-1.4185 I-0.2549 J0.3556
G03 X-1.2871 Y-1.3976 I-0.0249 J0.0025
G02 Y1.3976 I1.2871 J1.3976
G03 X-1.2791 Y1.4185 I-0.0169 J0.0184
G02 X-0.5889 Y1.817 I0.4354 J0.0429
G03 X-0.5669 Y1.8135 I0.0146 J0.0203
G02 X1.8539 Y0.4158 I0.5669 J-1.8135
G03 X1.868 Y0.3985 I0.0244 J0.0055
G02 X2.125 Y0 I-0.1805 J-0.3985
G03 X2.225 Y-0.1 I0.1 J0
G40 G01 X2.6
G113 (TURN OFF G112)
M155
G18 (RETURN TO XZ PLANE)
G00 Z3.
M135 (STOP LIVE TOOL)
G28 H0.
M30
%
```

G112 PROGRAM, SIMPLE AXIAL HEX

The following is a simple program, written for a mill, to cut a hex on the end of bar stock. The program takes the standard XY mill locations, and converts them to XC lathe moves with the use of a G112.

```
%
O000069 (.688in HEX ON .750IN STOCK)
G54
T101 (TOOL 1= .5IN EM)
M154
G00 C0.
G97 M133 P1500
G00 Z1.
G00 G98 X1.5 Z0.1 (Position)
G112               (XY TO XC INTERP)
G17                (Set to XY plane)
G00 X-0.5 Y0.594   (LEADIN)
G01 Z-0.5 F25.     (Z PLUNGE)
G01 X-0.343 F10.   (Point 1)
X0.343             (Point 2)
X0.6859 Y0.        (Point 3)
X0.343 Y-0.594     (Point 4)
X-0.343            (Point 5)
X-0.6859 Y0.       (Point 6)
X-0.343 Y0.594     (Point 1)
X-0.2853 Y0.594    (LEADOUT)
G113
G18                (Set to XZ plane)
G00 Z3.
M30
%
```

APPENDIX D.   LATHE TOOL PRESETTER CALIBRATION

Tools needed for calibration
    O.D. turning tool
    1 piece of stock that will fit in chuck jaws
    0-1.0" Micrometer used to measure LTP stylist
    Micrometer to inspect stock diameter

If you have a manual tool probe (LTP) skip to the calibration procedure below. If you have an automatic tool probe (ATP), **Verify the ATP arm operates correctly**;

Select **MDI/DNC** then key in M104; M105; and press insert.
In Single Block mode, press **Cycle Start** button to run M104 to bring probe down.
Press **Cycle Start** button to run M105 to bring the probe arm back up.

Calibration Procedure
1) Install an O.D. turning tool in station one of the turret and clamp a piece of material in the chuck.
2) Select **OFFSET** and clear the offset values for tool #1 in the **Tool Geometry** page
3) Use tool #1 and take a skim cut on the test cut material. Do not jog the X-Axis away from the cut diameter. Move the tool away from the stock by jogging in the Z+ direction.
4) Stop the spindle and use a micrometer to measure the diameter. Record cut diameter here _____.
5) Highlight X-Axis offset for tool #1 on the **Tool Geometry** page. Then press the **X DIA. MEAS** button and enter the measured diameter from above. Record the offset here as a positive number X _____ (Offset A).
6) Change the value of settings #59 thru #63 to zero.
7) Jog the tool away to a safe position, key in M104 in MDI mode, and press cycle start to bring the probe to the down position. With manual LTP, pull the tool arm to the down position.
8) Using the hand wheel, jog the Z-Axis so the tip of tool #1 is approximately centered on the probe in the Z-Axis. Jog the X-Axis until the tip is about .04" (1mm) above the probe in the X-Axis.
9) Select .001 jog increment and press **–X** button until probe sounds off and stops the tool. Record the offset here as a positive number X _____ (Offset B).
10) Subtract Offset B from Offset A and Record the value here _____. Enter this value into Setting #59.
11) We will skip setting #60 until step 14 below.
12) Enter zero as the value for setting #61.
13) Use a Micrometer to measure the probe width and record the value here _____. Enter this value for setting #62 and #63 as a positive number.
14) Multiply the probe width by two. Enter that number here _____. Now, take that value and subtract it from Setting #59. This is the number that you will enter into setting #60 _____ as a positive number.
15) The Lathe Tool Probe is now calibrated.

This page intentionally left blank.
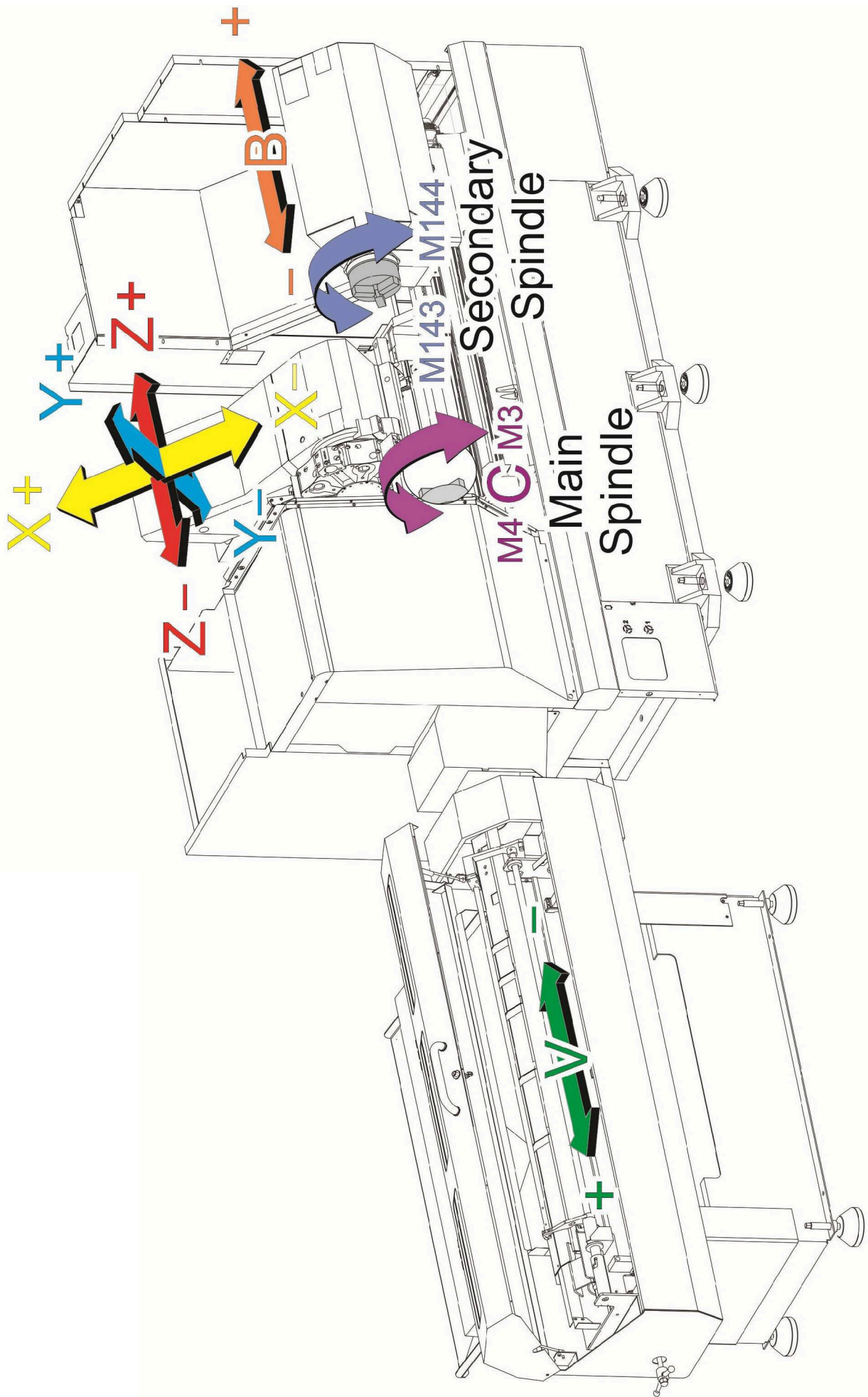
APPENDIX E.   PRINTABLE REFERENCE MATERIAL

The following pages contain helpful, printable reference material for the programmer.
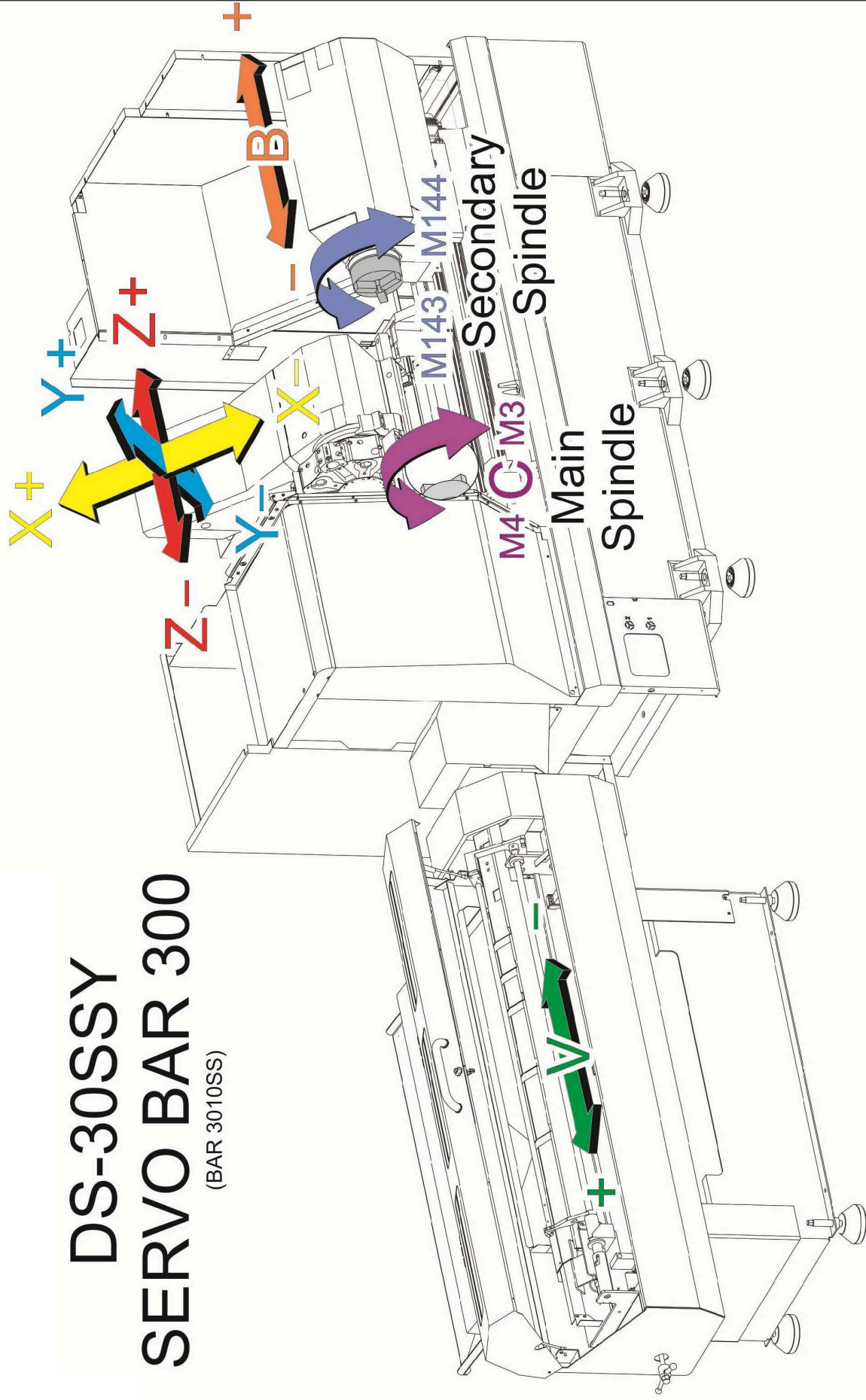
This page intentionally left blank.

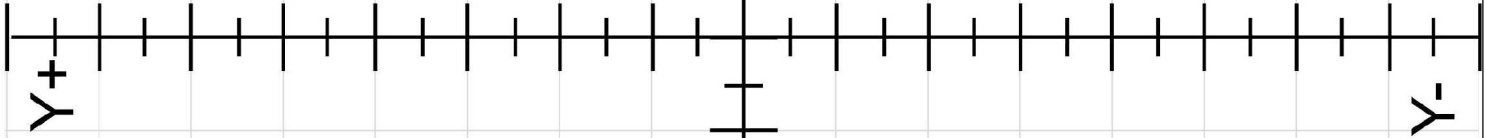*Haas Technical Publications*

This page intentionally left blank.

B +
B −
Z +
Y +
X +
X −
Y −
Z −
M143 M144
Secondary Spindle
M3
C
M4
Main Spindle
−
+

This page intentionally left blank.

DS-30SSY
SERVO BAR 300
(BAR 3010SS)

B +
B −

Z+
Y+
X+
X−
Y−
Z−

M144
M143
Secondary Spindle

M3
M4
C
Main Spindle

A −
A +

This page intentionally left blank.

Axial Milling
G17 (XY)Plane

ØX+

Y+

Y-

ØX-

ØX+

G17(XY)

Y-

ØX-

Y+
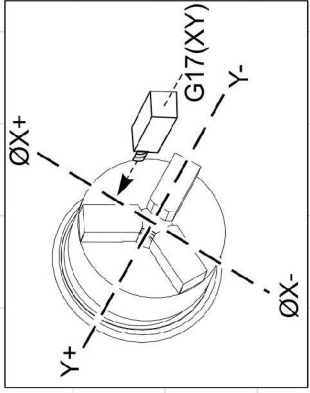
This page intentionally left blank.

Radial Milling
G19 (YZ)Plane

Z+

Y-

Y+

Z-

G19(YZ)

ØX+

Y-

ØX-

Y+

This page intentionally left blank.

Milling
G17 (XY)Plane

X+

Y+

Y-

X-

G17

This page intentionally left blank.