SHOP NOTES

**Pocket Guide and Reference Charts
for CNC Machinists**

– Made in the U.S.A. –

TABLE OF CONTENTS

Haas operator's manuals and other information may be downloaded for free from DIY.HaasCNC.com. Haas manuals and information are listed for you to access and download. Information is updated as necessary.

Decimal Equivalent Chart .0059 – .0980

Decimal Equiv.	Drill Size	mm	Tap Sizes	Decimal Equiv.	Drill Size	mm	Tap Sizes
.0059	97	0.150		.0320	67	0.813	
.0063	96	0.160		.0330	66	0.838	
.0067	95	0.170		.0350	65	0.889	
.0071	94	0.180		.0360	64	0.914	
.0075	93	0.191		.0370	63	0.940	
.0079	92	0.201		.0380	62	0.965	
.0083	91	0.211		.0390	61	0.991	
.0087	90	0.221		.0400	60	1.016	
.0091	89	0.231		.0410	59	1.041	
.0095	88	0.241		.0420	58	1.067	
.0100	87	0.254		.0430	57	1.092	
.0105	86	0.267		.0465	56	1.181	
.0110	85	0.279		.0469	$3/64$	1.191	#0-80
.0115	84	0.292		.0520	55	1.321	
.0120	83	0.305		.0550	54	1.397	
.0125	82	0.318		.0595	53	1.511	#1-64 • #1-72
.0130	81	0.330		.0625	$1/16$	1.588	
.0135	80	0.343		.0635	52	1.613	
.0145	79	0.368		.0670	51	1.702	
.0156	$1/64$	0.397		.0700	50	1.778	#2-56 • #2-64
.0160	78	0.406		.0730	49	1.854	
.0180	77	0.457		.0760	48	1.930	
.0200	76	0.508		.0781	$5/64$	1.984	
.0210	75	0.533		.0785	47	1.994	#3-48
.0225	74	0.572		.0810	46	2.057	
.0240	73	0.610		.0820	45	2.083	#3-56
.0250	72	0.635		.0860	44	2.184	
.0260	71	0.660		.0890	43	2.261	#4-40
.0280	70	0.711		.0935	42	2.375	#4-48
.0292	69	0.742		.0938	$3/32$	2.381	
.0310	68	0.787		.0960	41	2.438	
.0313	$1/32$	0.794		.0980	40	2.489	

Tap drill sizes above based on approximately 75% full thread
Tap Maj. thread dia.   #0 = .060   #1 = .073   #2 = .086   #3 = .099   #4 = .112
Tap # x .013 + .060 = Thread # OD

Decimal Equivalent Chart .0995 – .2969

Decimal Equiv.	Drill Size	mm	Tap Sizes	Decimal Equiv.	Drill Size	mm	Tap Sizes
.0995	39	2.527		.1875	$3/16$	4.763	#12-32
.1015	38	2.578	#5-40	.1890	12	4.801	
.1040	37	2.642	#5-44	.1910	11	4.851	
.1065	36	2.705	#6-32	.1935	10	4.915	
.1094	$7/64$	2.778		.1960	9	4.978	
.1100	35	2.794		.1990	8	5.055	
.1110	34	2.819		.2010	7	5.105	$1/4$-20
.1130	33	2.870	#6-40	.2031	$13/64$	5.159	
.1160	32	2.946		.2040	6	5.182	
.1200	31	3.048		.2055	5	5.220	
.1250	$1/8$	3.175		.2090	4	5.309	
.1285	30	3.264		.2130	3	5.410	$1/4$-28
.1360	29	3.454	#8-32 • #8-36	.2188	$7/32$	5.556	$1/4$-32
.1405	28	3.569		.2210	2	5.613	
.1406	$9/64$	3.572		.2280	1	5.791	
.1440	27	3.658		.2340	A	5.944	
.1470	26	3.734		.2344	$15/64$	5.953	
.1495	25	3.797	#10-24	.2380	B	6.045	
.1520	24	3.861		.2420	C	6.147	
.1540	23	3.912		.2460	D	6.248	
.1563	$5/32$	3.969		.2500	$1/4$&E	6.350	
.1570	22	3.988		.2570	F	6.528	$5/16$-18
.1590	21	4.039	#10-32	.2610	G	6.629	
.1610	20	4.089		.2656	$17/64$	6.747	
.1660	19	4.216		.2660	H	6.756	
.1695	18	4.305		.2720	I	6.909	$5/16$-24
.1719	$11/64$	4.366		.2770	J	7.036	
.1730	17	4.394		.2810	K	7.137	
.1770	16	4.496	#12-24	.2813	$9/32$	7.144	$5/16$-32
.1800	15	4.572		.2900	L	7.366	
.1820	14	4.623	#12-28	.2950	M	7.493	
.1850	13	4.699		.2969	$19/64$	7.541	

Tap drill sizes above based on approximately 75% full thread
Tap Maj. thread dia.   #5 = .125   #6 = .138   #8 = .164   #10 = .190   #12 = .216
Tap # x .013 + .060 = Thread # OD

Decimal Equiv.	Drill Size	mm	Tap Sizes
.3020	N	7.671	
.3125	$5/16$	7.938	$3/8$-16
.3160	O	8.026	
.3230	P	8.204	
.3281	$21/64$	8.334	
.3320	Q	8.433	$3/8$-24
.3390	R	8.611	
.3438	$11/32$	8.731	$3/8$-32
.3480	S	8.839	
.3580	T	9.093	
.3594	$23/64$	9.128	
.3680	U	9.347	$7/16$-14
.3750	$3/8$	9.525	
.3770	V	9.576	
.3860	W	9.804	
.3906	$25/64$	9.922	$7/16$-20
.3970	X	10.084	
.4040	Y	10.262	$7/16$-28
.4063	$13/32$	10.319	
.4130	Z	10.490	
.4219	$27/64$	10.716	$1/2$-13
.4375	$7/16$	11.113	
.4531	$29/64$	11.509	$1/2$-20
.4688	$15/32$	11.906	$1/2$-28
.4844	$31/64$	12.303	$9/16$-12
.5000	$1/2$	12.700	$9/16$-18
.5156	$33/64$	13.097	$9/16$-24
.5313	$17/32$	13.494	$5/8$-11
.5469	$35/64$	13.891	

Decimal Equiv.	Drill Size	mm	Tap Sizes
.5625	$9/16$	14.288	$5/8$-18
.5781	$37/64$	14.684	$5/8$-24
.5938	$19/32$	15.081	
.6094	$39/64$	15.478	$11/16$-12
.6250	$5/8$	15.875	
.6406	$41/64$	16.272	$11/16$-20 • $11/16$-24
.6563	$21/32$	16.669	$3/4$-10
.6719	$43/64$	17.066	
.6875	$11/16$	17.462	$3/4$-16
.7031	$45/64$	17.859	$3/4$-20
.7188	$23/32$	18.256	
.7344	$47/64$	18.653	$13/16$-12
.7500	$3/4$	19.050	$13/16$-16
.7656	$49/64$	19.447	$13/16$-20 • $7/8$-9
.7813	$25/32$	19.844	
.7969	$51/64$	20.241	$7/8$-14
.8125	$13/16$	20.637	
.8281	$53/64$	21.034	$7/8$-20
.8438	$27/32$	21.431	
.8594	$55/64$	21.828	$15/16$-12
.8750	$7/8$	22.225	$15/16$-16 • 1.0-8
.8906	$57/64$	22.622	$15/16$-20
.9063	$29/32$	23.019	
.9219	$59/64$	23.416	1.0-12
.9375	$15/16$	23.813	
.9531	$61/64$	24.209	1.0-20
.9688	$31/32$	24.606	
.9844	$63/64$	25.003	
1.000	1	25.400	

Tap Thread Size	Approx. inside Dia.	Approx. outside Dia.	Tap Drill
$1/8$ – 27	$1/4$	$3/8$	$11/32$
$1/4$ – 18	$3/8$	$17/32$	$7/16$
$3/8$ – 18	$1/2$	$11/16$	$37/64$
$1/2$ – 14	$5/8$	$13/16$	$23/32$
$3/4$ – 14	$13/16$	1	$59/64$
1 – $11 1/2$	$1 1/16$	$1 5/16$	$1 5/32$
$1 1/4$ – $11 1/2$	$1 3/8$	$1 5/8$	$1 1/2$
$1 1/2$ – $11 1/2$	$1 5/8$	$1 7/8$	$1 47/64$
2 – $11 1/2$	$2 1/16$	$2 3/8$	$2 7/32$
$2 1/2$ – 8	$2 9/16$	$2 7/8$	$2 5/8$

Pipe sizes are generally determined by the inside diameter of the pipe. The chart above gives nominal and approximate actual dimensions of commonly used sizes of standard threaded pipe.

Inch/Metric Conversion: Inches = mm x 25.4 | mm = Inches ÷ 25.4

Tap drill sizes above based on approximately 75% full thread
A decimal equivalent chart can be displayed on a Haas control by pressing the HELP/CALC button, and then selecting the Drill Table tab. Use the jog handle or cursor keys to scroll through the chart. (Classic control only.)

Spindle Command: You can stop or start the spindle with CW or CCW (FWD and REV on a lathe) any time you're at a Single Block stop or a Feed Hold. When you restart the program with CYCLE START, the spindle will be turned back on to the previously defined speed.

MILLIMETER TO INCH CHART 0.01 – 12.5

mm	Inch	mm	Inch	mm	Inch	mm	Inch
0.01	.0004	2.6	.1024	6.0	.2362	9.4	.3701
0.02	.0008	2.7	.1063	6.1	.2402	9.5	.3740
0.03	.0012	2.8	.1102	6.2	.2441	9.6	.3780
0.04	.0016	2.9	.1142	6.3	.2480	9.7	.3819
0.05	.0020	3.0	.1181	6.4	.2520	9.8	.3858
0.06	.0024	3.1	.1220	6.5	.2559	9.9	.3898
0.07	.0028	3.2	.1260	6.6	.2598	10.0	.3937
0.08	.0032	3.3	.1299	6.7	.2638	10.1	.3976
0.09	.0035	3.4	.1339	6.8	.2677	10.2	.4016
0.1	.0039	3.5	.1378	6.9	.2717	10.3	.4055
0.2	.0079	3.6	.1417	7.0	.2756	10.4	.4094
0.3	.0118	3.7	.1457	7.1	.2795	10.5	.4134
0.4	.0157	3.8	.1496	7.2	.2835	10.6	.4173
0.5	.0197	3.9	.1535	7.3	.2874	10.7	.4213
0.6	.0236	4.0	.1575	7.4	.2913	10.8	.4252
0.7	.0276	4.1	.1614	7.5	.2953	10.9	.4291
0.8	.0315	4.2	.1654	7.6	.2992	11.0	.4331
0.9	.0354	4.3	.1693	7.7	.3031	11.1	.4370
1.0	.0394	4.4	.1732	7.8	.3071	11.2	.4409
1.1	.0433	4.5	.1772	7.9	.3110	11.3	.4449
1.2	.0472	4.6	.1811	8.0	.3150	11.4	.4488
1.3	.0512	4.7	.1850	8.1	.3189	11.5	.4528
1.4	.0551	4.8	.1890	8.2	.3228	11.6	.4567
1.5	.0591	4.9	.1929	8.3	.3268	11.7	.4606
1.6	.0630	5.0	.1969	8.4	.3307	11.8	.4646
1.7	.0669	5.1	.2008	8.5	.3346	11.9	.4685
1.8	.0709	5.2	.2047	8.6	.3386	12.0	.4724
1.9	.0748	5.3	.2087	8.7	.3425	12.1	.4764
2.0	.0787	5.4	.2126	8.8	.3465	12.2	.4803
2.1	.0827	5.5	.2165	8.9	.3504	12.3	.4843
2.2	.0866	5.6	.2205	9.0	.3543	12.4	.4882
2.3	.0906	5.7	.2244	9.1	.3583	12.5	.4921
2.4	.0945	5.8	.2283	9.2	.3622		
2.5	.0984	5.9	.2323	9.3	.3661		

MILLIMETER TO INCH CHART 12.6 – 25.4

mm	Inch	mm	Inch	mm	Inch	mm	Inch
12.6	.4961	15.9	.6260	19.2	.7559	22.5	.8858
12.7	.5000	16.0	.6299	19.3	.7598	22.6	.8898
12.8	.5039	16.1	.6339	19.4	.7638	22.7	.8937
12.9	.5079	16.2	.6378	19.5	.7677	22.8	.8976
13.0	.5118	16.3	.6417	19.6	.7717	22.9	.9016
13.1	.5157	16.4	.6457	19.7	.7756	23.0	.9055
13.2	.5197	16.5	.6496	19.8	.7795	23.1	.9094
13.3	.5236	16.6	.6535	19.9	.7835	23.2	.9134
13.4	.5276	16.7	.6575	20.0	.7874	23.3	.9173
13.5	.5315	16.8	.6614	20.1	.7913	23.4	.9213
13.6	.5354	16.9	.6654	20.2	.7953	23.5	.9252
13.7	.5394	17.0	.6693	20.3	.7992	23.6	.9291
13.8	.5433	17.1	.6732	20.4	.8031	23.7	.9331
13.9	.5472	17.2	.6772	20.5	.8071	23.8	.9370
14.0	.5512	17.3	.6811	20.6	.8110	23.9	.9409
14.1	.5551	17.4	.6850	20.7	.8150	24.0	.9449
14.2	.5591	17.5	.6890	20.8	.8189	24.1	.9488
14.3	.5630	17.6	.6929	20.9	.8228	24.2	.9528
14.4	.5669	17.7	.6968	21.0	.8268	24.3	.9567
14.5	.5709	17.8	.7008	21.1	.8307	24.4	.9606
14.6	.5748	17.9	.7047	21.2	.8346	24.5	.9646
14.7	.5787	18.0	.7087	21.3	.8386	24.6	.9685
14.8	.5827	18.1	.7126	21.4	.8425	24.7	.9724
14.9	.5866	18.2	.7165	21.5	.8465	24.8	.9764
15.0	.5906	18.3	.7205	21.6	.8504	24.9	.9803
15.1	.5945	18.4	.7244	21.7	.8543	25.0	.9843
15.2	.5984	18.5	.7283	21.8	.8583	25.1	.9882
15.3	.6024	18.6	.7323	21.9	.8622	25.2	.9921
15.4	.6063	18.7	.7362	22.0	.8661	25.3	.9961
15.5	.6102	18.8	.7402	22.1	.8701	25.4	1.0
15.6	.6142	18.9	.7441	22.2	.8740		
15.7	.6181	19.0	.7480	22.3	.8780		
15.8	.6220	19.1	.7520	22.4	.8819		

Setting 9 on a Haas allows you to change between **inch** and **millimeter dimensioning**.

When in EDIT or MEM mode, you can select and **display another program** from Memory quickly by entering the program number (Onnnnn) and pressing the down arrow key.

METRIC TAPS

METRIC THREAD PITCH CONVERSION

Tap Sizes	MM Tap Drill	Drill Dia. in Inches	Tap Sizes	MM Tap Drill	Drill Dia. in Inches
M1 x 0.25	0.75	.0295	M14 x 2	12.00	.4724
M1.1 x 0.25	0.85	.0335	M14 x 1.5	12.50	.4921
M1.2 x 0.25	0.95	.0374	M16 x 2	14.00	.5512
M1.4 x 0.3	1.10	.0433	M16 x 1.5	14.50	.5709
M1.6 x 0.35	1.25	.0492	M18 x 2.5	15.50	.6102
M1.8 x 0.35	1.45	.0571	M18 x 1.5	16.50	.6496
M2 x 0.4	1.60	.0630	M20 x 2.5	17.50	.6890
M2.2 x 0.45	1.75	.0689	M20 x 1.5	18.50	.7283
M2.5 x 0.45	2.05	.0807	M22 x 2.5	19.50	.7677
M3 x 0.5	2.50	.0984	M22 x 1.5	20.50	.8071
M3.5 x 0.6	2.90	.1142	M24 x 3	21.00	.8268
M4 x 0.7	3.30	.1299	M24 x 2	22.00	.8661
M4.5 x 0.75	3.70	.1457	M27 x 3	24.00	.9449
M5 x 0.8	4.20	.1654	M27 x 2	25.00	.9843
M6 x 1	5.00	.1969	M30 x 3.5	26.50	1.0433
M7 x 1	6.00	.2362	M30 x 2	28.00	1.1024
M8 x 1.25	6.75	.2657	M33 x 3.5	29.50	1.1614
M8 x 1	7.00	.2756	M33 x 2	31.00	1.2205
M10 x 1.5	8.50	.3346	M36 x 4	32.00	1.2598
M10 x 1.25	8.75	.3445	M36 x 3	33.00	1.2992
M12 x 1.75	10.20	.4016	M39 x 4	35.00	1.3780
M12 x 1.25	10.80	.4252	M39 x 3	36.00	1.4173

Metric Thd. Pitch	Thd. Pitch in Inches	Threads Per In.	Basic Height
. 25	. 00984	101.6002	. 00639
. 30	. 01181	84.6668	. 00767
. 35	. 01378	72.5716	. 00895
. 40	. 01575	63.5001	. 01023
. 45	. 01772	56.4446	. 01151
. 50	. 01969	50.8001	. 01279
. 60	. 02362	42.3334	. 01534
. 70	. 02756	36.2858	. 01790
. 75	. 02953	33.8667	.01918
. 80	. 03150	31.7501	.02046
. 90	. 03543	28.2228	. 02301
1.00	. 03937	25.4000	.02557
1.25	. 04921	20.3200	. 03196
1.50	. 05906	16.9334	. 03836
1.75	. 06890	14.5143	. 04475
2.00	. 07874	12.7000	. 05114
2.50	. 09843	10.1600	. 06393
3.00	. 11811	8.4667	. 07671
3.50	. 13780	7.2572	. 08950
4.00	. 15748	6.3500	. 10229
4.50	. 17717	5.6445	. 11508
5.00	. 19685	5.0800	. 12785
6.00	. 23622	4.2333	. 15344

Tap drill sizes based on 75% full metric thread
Metric tap and drill sizes can be displayed on a Haas control by pressing the HELP/CALC button twice, and then selecting the Drill Table tab. (Classic control only.)

Clearing Current Commands Values: On a Haas, the values in the CURNT COMDS display pages for Tool Life, Tool Load, and Timer registers can be cleared by cursor-selecting the one you wish to clear and pressing ORIGIN. To clear everything in a column, cursor to the top of that column (onto the title) and press ORIGIN.

HAAS MILL G-CODES (may vary with software version)

Code	Description	Group
G00*	Rapid Motion Positioning	01
G01	Linear Interpolation Motion	01
G02	Circular Interpolation Motion CW	01
G03	Circular Interpolation Motion CCW	01
G04	Dwell	00
G09	Exact Stop	00
G10	Set Offsets	00
G12	Circular Pocket Milling CW	00
G13	Circular Pocket Milling CCW	00
G17*	XY Plane Selection	02
G18	XZ Plane Selection	02
G19	YZ Plane Selection	02
G20	Select Inches	06
G21	Select Metric	06
G28	Return To Machine Zero Point	00
G29	Return From Reference Point	00
G31	Feed Until Skip	00
G35	Automatic Tool Diameter Measurement	00
G36	Automatic Work Offset Measurement	00
G37	Automatic Tool Offset Measurement	00
G40*	Cutter Compensation Cancel	07
G41	2D Cutter Compensation Left	07
G42	2D Cutter Compensation Right	07
G43	Tool Length Compensation + (Add)	08
G44	Tool Length Compensation - (Subtract)	08
G47	Text Engraving	00
G49*	G43/G44/G143 Cancel	08
G50*	Cancel Scaling	11
G51	Scaling	11
G52	Set Work Coordinate System	00 or 12
G53	Non-Modal Machine Coordinate Selection	00

* default †Classic control only

Complete descriptions of all Haas G- and M-codes are available at DIY.HaasCNC.com.

HAAS MILL G-CODES

Code	Description	Group
G54*	Select Work Coordinate System #1	12
G55	Select Work Coordinate System #2	12
G56	Select Work Coordinate System #3	12
G57	Select Work Coordinate System #4	12
G58	Select Work Coordinate System #5	12
G59	Select Work Coordinate System #6	12
G60	Uni-Directional Positioning	00
G61	Exact Stop Mode	15
G64*	G61 Cancel	15
G65	Macro Subroutine Call Option	00
G68	Rotation	16
G69*	Cancel G68 Rotation	16
G70	Bolt Hole Circle	00
G71	Bolt Hole Arc	00
G72	Bolt Holes Along an Angle	00
G73	High-Speed Peck Drilling Canned Cycle	09
G74	Reverse Tap Canned Cycle	09
G76	Fine Boring Canned Cycle	09
G77	Back Bore Canned Cycle	09
G80*	Canned Cycle Cancel	09
G81	Drill Canned Cycle	09
G82	Spot Drill Canned Cycle	09
G83	Normal Peck Drilling Canned Cycle	09
G84	Tapping Canned Cycle	09
G85	Boring Canned Cycle	09
G86	Bore and Stop Canned Cycle	09
G87†	Bore In and Manual Retract Canned Cycle	09
G88†	Bore In, Dwell, Manual Retract Canned Cycle	09
G89	Bore In, Dwell, Bore Out Canned Cycle	09
G90*	Absolute Position Command	03
G91	Incremental Position Command	03

* default †Classic control only

In the Offset display on a Haas, you can **zero all offsets** at once by pressing ORIGIN, and following the simple on-screen commands. **You can't undo this.**

HAAS MILL G-CODES (may vary with software version)

Code	Description	Group
G92	Set Work Coordinate Systems Shift Value	00
G93	Inverse Time Feed Mode	05
G94*	Feed Per Minute Mode	05
G95	Feed per Revolution	05
G98*	Canned Cycle Initial Point Return	10
G99	Canned Cycle R Plane Return	10
G100	Cancel Mirror Image	00
G101	Enable Mirror Image	00
G102	Programmable Output to RS-232	00
G103	Limit Block Buffering	00
G107	Cylindrical Mapping	00
G110-G129	Coordinate System #7 - #26	12
G136	Automatic Work Offset Center Measurement	00
G141	3D+ Cutter Compensation	07
G143	5-Axis Tool Length Compensation +	08
G150	General Purpose Pocket Milling	00
G153	5-Axis High Speed Peck Drilling Canned Cycle	09
G154	Select Work Coordinates P1-P99	12
G155	5-Axis Reverse Tap Canned Cycle	09
G161	5-Axis Drill Canned Cycle	09
G162	5-Axis Spot Drill Canned Cycle	09
G163	5-Axis Normal Peck Drilling Canned Cycle	09
G164	5-Axis Tapping Canned Cycle	09
G165	5-Axis Boring Canned Cycle	09

* default

You can edit programs on a Haas while a program is running, using **Background (BG) Edit**. When running a program in MEM mode from the Program display, hit the EDIT button until the Background Editor pane appears on the right side of the screen. Press SELECT PROG to see a list of Memory programs you can BG Edit. Changes will take effect the next time the program is opened. (Classic control only.)

To background edit on an NGC machine, while a program is running, press **LIST PROG**, highlight the program you'd like to background edit, and press **ALTER**.

Code	Description	Group
G166	5-Axis Bore and Stop Canned Cycle	09
G169	5-Axis Bore and Dwell Canned Cycle	09
G174	CCW Non-Vertical Rigid Tap	00
G184	CW Non-Vertical Rigid Tap	00
G187	Setting the Smoothness Level	00
G188	Get Program From PST	00
G234	Tool Center Point Control (TCPC)	08
G254	Dynamic Work Offset (DWO)	23
G255	Cancel Dynamic Work Offset (DWO)	23

To Zero the POS-OPER Display: This display is used for reference only. Each axis can be zeroed out independently, to then show its position relative to where you selected to zero that axis. To zero out a specific axis, press HAND JOG, and then press POSIT. When you Handle Jog the X, Y, or Z axis and then press ORIGIN, the axis that is selected will be zeroed. Or, you can press an X, Y, or Z letter key and then ORIGIN to zero that axis display. You can also press the X, Y, or Z key and enter a number (X2.125), then press ORIGIN to enter the number in that axis display.

HAAS MILL M-CODES (may vary with software version)

M00	Stop Program
M01	Optional Program Stop
M02	Program End
M03	Spindle Commands
M04	Spindle Commands
M05	Spindle Commands
M06	Tool Change
M07	Shower Coolant On
M08	Coolant On
M09	Coolant Off
M10	Engage 4th Axis Brake
M11	Release 4th Axis Brake
M12	Engage 5th Axis Brake
M13	Release 5th Axis Brake
M16	Tool Change
M17	Unclamp APC Pallet and Open APC Door
M18	Clamp APC Pallet and Close Door
M19	Orient Spindle
M21-M28	Optional User M Function with M-Fin
M30	Program End and Reset
M31	Chip Conveyor Forward
M33	Chip Conveyor Stop
M34	Coolant Increment
M35	Coolant Decrement
M36	Pallet Part Ready
M39	Rotate Tool Turret
M41	Low Gear Override
M42	High Gear Override
M46	Jump if Pallet Loaded
M48	Check Validity of Current Program

When **Setting 32** on a Haas machine is set to **IGNORE**, then all commands for turning coolant on or off will be ignored. The coolant can still be turned on and off manually with the COOLNT button.

HAAS MILL M-CODES

M49	Set Status of Pallet
M50	Execute Pallet Change
M51-M58	Set Optional User M-codes
M59	Set Output Relay
M61-M68	Clear Optional User M-codes
M69	Clear Output Relay
M73	Tool Air Blast On
M74	Tool Air Blast Off
M75	Set G35 or G136 Reference Point
M76	Control Display Inactive
M77	Control Display Active
M78	Alarm if Skip Signal Found
M79	Alarm if Skip Signal Not Found
M80	Auto Door Open
M81	Auto Door Close
M82	Tool Unclamp
M83	Auto Air Gun On
M84	Auto Air Gun Off
M86	Tool Clamp
M88	Through-Spindle Coolant On
M89	Through-Spindle Coolant Off
M95	Sleep Mode
M96	Jump If No Input
M97	Local Sub-Program Call
M98	Sub-Program Cal
M99	Sub-Program Return or Loop
M109	Interactive User Input
M130	Display Media
M131	Cancel Display Media
M138	Spindle Speed Variation On
M139	Spindle Speed Variation Off

Jog Keys: You can select an axis for jogging on a Haas by entering the axis letter on the input line and then pressing the HANDLE JOG button.

Code	Description	Group
G00*	Rapid Motion Positioning	01
G01	Linear Interpolation Motion	01
G02	CW Circular Interpolation Motion	01
G03	CCW Circular Interpolation Motion	01
G04	Dwell	00
G09	Exact Stop	00
G10	Set Offsets	00
G14	Secondary Spindle Swap	17
G15	Secondary Spindle Cancel	17
G17	XY Plane Selection	00
G18*	XZ Plane Selection	02
G19	YZ Plane Selection	02
G20	Select Inches	06
G21	Select Metric	06
G28	Return To Machine Zero Point	00
G29	Return From Reference Point	00
G31	Skip Function	00
G32	Thread Cutting	01
G40*	Tool Nose Compensation Cancel	07
G41	Tool Nose Compensation (TNC) Left	07
G42	Tool Nose Compensation (TNC) Right	07
G50	Set Global coordinate Offset FANUC, YASNAC	00
G51	Cancel Offset (YASNAC)	00
G52	Set Local Coordinate System FANUC	00
G53	Machine Coordinate Selection	00
G54*	Coordinate System #1 FANUC	12
G55	Coordinate System #2 FANUC	12
G56	Coordinate System #3 FANUC	12
G57	Coordinate System #4 FANUC	12
G58	Coordinate System #5 FANUC	12

* default

Code	Description	Group
G59	Coordinate System #6 FANUC	12
G61	Exact Stop Modal	15
G64*	Exact Stop Cancel G61	15
G65	Macro Subroutine Call Option	00
G70	Finishing Cycle	00
G71	O.D./I.D. Stock Removal Cycle	00
G72	End Face Stock Removal Cycle	00
G73	Irregular Path Stock Removal Cycle	00
G74	End Face Grooving Cycle	00
G75	O.D./I.D. Grooving Cycle	00
G76	Threading Cycle, Multiple Pass	00
G80*	Canned Cycle Cancel	09
G81	Drill Canned Cycle	09
G82	Spot Drill Canned Cycle	09
G83	Normal Peck Drilling Canned Cycle	09
G84	Tapping Canned Cycle	09
G85	Boring Canned Cycle	09
G86	Bore and Stop Canned Cycle	09
G87	Bore and Manual Retract Canned Cycle	09
G88	Bore and Dwell and Manual Retract Canned Cycle	09
G89	Bore and Dwell Canned Cycle	09
G90	O.D./I.D. Turning Cycle	01
G92	Threading Cycle	01
G94	End Facing Cycle	01
G95	Live Tooling Rigid Tap (Face)	09
G96	Constant Surface Speed On	13
G97*	Constant Surface Speed Off	13
G98	Feed Per Minute	10
G99*	Feed Per Revolution	10
G100	Disable Mirror Image	00

* default

Setting 22 on a Haas, **Can Cycle Delta Z**, defines the distance above the previous peck that a tool will rapid back to during a mill and lathe G83 peck drill or the amount it pulls back in a G74 and G75 lathe grooving cycle. It also defines the distance the tool retracts to break the chip in a mill G73 peck drill canned cycle.

HAAS LATHE G-CODES (may vary with software version)

Code	Description	Group
G101	Enable Mirror Image	00
G102	Programmable Output to RS-232	00
G103	Limit Block Lookahead	00
G105	Servo Bar Command	09
G110	Coordinate System #7	12
G111	Coordinate System #8	12
G112	XY to XC interpretation	04
G113	Cancel G112	04
G114-G129	Coordinate System #9 - #24	12
G154	Select Work Coordinates P1-99	12
G159	Background Pickup / Part Return	
G160	APL Axis Command Mode Only	
G161	APL Axis Command Mode Off	
G184	Reverse Tapping Canned Cycle For Left Hand Threads	09
G186	Reverse Live Tool Rigid Tap (For Left Hand Threads)	10
G187	Accuracy Control	00
G195	Forward Live Tool Radial Tapping (Diameter)	00
G196	Reverse Live Tool Radial Tapping (Diameter)	00
G198	Disengage Synchronous Spindle Control	00
G199	Engage Synchronous Spindle Control	00
G200	Index on the Fly	00
G211	Manual Tool Setting	
G212	Auto Tool Setting	
G241	Radial Drill Canned Cycle	09
G242	Radial Spot Drill Canned Cycle	09
G243	Radial Normal Peck Drilling Canned Cycle	09
G245	Radial Boring Canned Cycle	09
G246	Radial Bore and Stop Canned Cycle	09
G247	Radial Bore and Manual Retract Canned Cycle	09
G248	Radial Bore and Dwell and Manual Retract Canned Cycle	09
G249	Radial Bore and Dwell Canned Cycle	09

Transferring Simple Calculations: In the Haas Calculator display, the number in the simple calculator box (upper left corner) can be transferred to any cursor-selected data line on the page in either EDIT or MDI. Cursor to the register to which you wish to transfer the calculator number, and press F3.

HAAS LATHE M-CODES

M00	Stop Program	
M01	Stop Program	
M02	Program End	
M03	Spindle On Fwd	
M04	Spindle On Rev	
M05	Spindle Stop	
M08	Coolant On	
M09	Coolant Off	
M10	Chuck Clamp	
M11	Chuck Unclamp	
M12	Auto Jet Air Blast On (Optional)	
M13	Auto Jet Air Blast Off (Optional)	
M14	Main Spindle Brake On (Optional C-Axis)	
M15	Main Spindle Brake Off (Optional C-Axis)	
M17	Turret Rotation Fwd	
M18	Turret Rotation Rev	
M19	Orient Spindle (Optional)	
M21	Tailstock Advance (Optional)	
M22	Tailstock Retract (Optional)	
M23	Chamfer Out of Thread On	
M24	Chamfer Out of Thread Off	
M30	End of Program and Reset	
M31	Chip Auger Forward (Optional)	
M33	Chip Auger Stop (Optional)	
M36	Parts Catcher On (Optional)	
M37	Parts Catcher Off (Optional)	
M38	Spindle Speed Variation On	
M39	Spindle Speed Variation Off	
M41	Low Gear (Optional)	
M42	High Gear (Optional)	

On a Haas, you can use the **DIST-TO-GO screen to quickly zero** out the Position display for a reference move. When in Handle Jog mode and in the Position display, press any other operation mode key (EDIT, MEM, etc.), and then go back to Handle Jog. This will zero out all axes on the DIST-TO-GO display, and begin showing the distance moved.

HAAS LATHE M-CODES (may vary with software version)

M43	Turret Unlock (Service Use Only)
M44	Turret Lock (Service Use Only)
M51-M58	User M Turn On (Optional)
M59	Set Output Relay
M61-M68	User M Turn Off (Optional)
M69	Clear Output Relay
M76	Display Disable
M77	Display Enable
M78	Alarm if Skip Signal Found
M79	Alarm if Skip Signal Not Found
M85	Automatic Door Open (Optional)
M86	Automatic Door Close (Optional)
M88	High-Pressure Coolant On (Optional)
M89	High-Pressure Coolant Off (Optional)
M95	Sleep Mode
M96	Jump If No Signal
M97	Local Subprogram Call
M98	Subprogram Call
M99	Subprogram Return Or Loop

HAAS LATHE M-CODES

M104	Probe Arm Extend (Optional)
M105	Probe Arm Retract (Optional)
M109	Interactive User Input
M110	Secondary Spindle Chuck Clamp (Optional)
M111	Secondary Spindle Chuck Unclamp (Optional)
M112	Secondary Spindle Air Blast On (Optional)
M113	Secondary Spindle Air Blast Off (Optional)
M114	Secondary Spindle Brake On (Optional)
M115	Secondary Spindle Brake Off (Optional)
M119	Secondary Spindle Orient (Optional)
M121-M128	User M-codes (Optional)
M130	Display Media
M131	Cancel Display Media
M133	Live Tool Fwd (Optional)
M134	Live Tool Rev (Optional)
M135	Live Tool Stop (Optional)
M143	Secondary Spindle Forward (Optional)
M144	Secondary Spindle Reverse (Optional)
M145	Secondary Spindle Stop (Optional)
M154	C-Axis Engage (Optional)
M155	C-Axis Disengage (Optional)

On a Haas, it's easy to **transfer a program from MDI** and save it to your list of programs. In the MDI display, make sure that the cursor is at the beginning of the MDI program. Enter a program number (Onnnnn) that's not being used. Then press ALTER and this will transfer the MDI data into your **List of Programs** under that program number.

To Rapid an Axis Home: You can rapid *all* axes to machine zero by pressing the **HOME G28** key. You can also send just one axis (X, Y, Z, A, or B) to machine zero in rapid motion. Enter the letter X, Y, Z, A, or B, then press **HOME G28** and that axis alone will rapid home. *CAUTION!* There is no warning to alert you of any possible collision!

ABBREVIATIONS & MEASUREMENT UNITS

°C = Degrees Celsius

DIA = Diameter

d = Depth of Cut

F = Feed in Inches or mm Per Minute (F)

°F = Degrees Fahrenheit

FPR = Feed Per Revolution (F)

FPT = Feed Per Tooth

IPM = Inches Per Minute

IPR = Inches Per Revolution

L = Length of Cut

MRR = Metal Removal Rate (cubic in./min.)

RPM = Revolutions Per Minute

SFM = Surface Feed Per Minute

SMPM = Surface Meters Per Minute

MMPM = Millimeters Per Minute

MMPR = Millimeters Per Revolution

T = Number of Teeth in a Cutter

TCm = Time Cutting in Minutes

TCs = Time Cutting in Seconds

TPI = Threads Per Inch

W = Width of Cut

Chip Conveyor - The chip conveyor on a Haas can be turned on or off when a program is running, either manually using the control keys or in the program using M-codes. The M-code equivalent to CHIP FWD is M31, and CHIP STOP is M33. You can set the Conveyor Cycle time (in minutes) with Setting 114, and the Conveyor On-Time (in minutes) with Setting 115.

MILL AND LATHE FORMULAS

Cutting Speed (surface feet/min.)
SFM = 0.262 x DIA x RPM

Revolutions Per Minute
RPM = 3.82 x SFM ÷ DIA

Feed Rate (in/min.)
IPM = FPT x T x RPM

Feed Per Revolution
FPR = IPM ÷ RPM

Feed Per Tooth (in)
FPT = IPM ÷ (RPM x T)

Metal Removal Rate
MRR = W x d x F

Converting IPR to IPM
IPM = IPR x RPM

Converting IPM to IPR
IPR = IPM ÷ RPM

Converting SFM to SMPM
SMPM = SFM x .3048

Converting IPR to MMPR
MMPR = IPR x 25.40

Distance over Time (in minutes)
L = IPM x TCm

Time Cutting over Distance (Mill) (minutes)
TCm = L ÷ IPM

Time Cutting over Distance (Mill) (seconds)
TCs = L ÷ IPM x 60

Time Cutting over Distance (Lathe) (seconds)
TCs = L ÷ (IPR x RPM) x 60

INCH METRIC CONVERSION

mm x 0.03937 = in.	in. x 25.4 = mm
m x 39.37 = in.	in. x 0.0254 = m
m x 3.2808 = ft	ft x 0.3048 = m
m x 1.0936 = yd	yd x 0.9144 = m
km x 0.621 = mi	mi x 1.6093 = km
Celsius to Fahrenheit (˚C x 1.8) + 32 = ˚F	Fahrenheit to Celsius (˚F - 32) ÷ 1.8 = ˚C

Setting 36 PROGRAM RESTART: When it is ON, you are able to start a program from the middle of a tool sequence. You cursor to the line on which you want to start and press CYCLE START. It will scan the entire program to ensure the tools, offsets, G codes, and axes positions are set correctly before starting and continuing at the block where the cursor is positioned. Although you can leave this setting ON all the time, it may cause the machine to perform certain activities unnecessarily, so it's best to turn it OFF when you're done using it.

TAPPING AND THREADING FORMULAS

INCH TAPS

Cut Tap Drill Size (inch) $=$ Thread Diameter $-$ $\dfrac{0.01299 \times \% \text{ of Full Thread}}{\text{Number of TPI}}$

% of Full Thread (inch) $=$ Number of TPI $\times$ $\dfrac{\text{Major DIA of Thread} - \text{Drilled DIA}}{0.01299}$

IPM (Mill Tapping Feed Rate) $=$ RPM $\div$ TPI

IPR (Lathe Threading) $=$ $1 \div$ TPI

Form Tap Drill Size $=$ Basic Tap DIA $-$ $\dfrac{0.0068 \times \% \text{ of Full Thread}}{\text{Number of TPI}}$

Recommended 65% form thread:
Form Tap Drill Size $=$ Basic Tap DIA $-$ $\dfrac{0.442}{\text{Number of TPI}}$

METRIC TAPS

Cut Tap Drill Size (metric) $=$ Thread Diameter [mm] $-$ $\dfrac{\% \text{ of Full Thread} \times \text{Pitch [mm]}}{76.98}$

% of Full Thread (metric) $=$ $\dfrac{76.98}{\text{MM Pitch}}$ $\times$ (Thread DIA [mm] $-$ Drilled Hole DIA [mm])

MMPM $=$ RPM $\times$ Metric Pitch
MMPR $=$ Pitch [mm]
Recommended 65% form thread:
Form Tap Drill Size (metric) $=$ Basic Tap DIA $-$ $\dfrac{65 \times \text{Pitch [mm]}}{147.06}$

Tap Calculator: Try the new tap calculator on the next gen control.
CURNT COMDS > CALCULATOR > TAPPING

TAP DRILL CALCULATION

FIND TAP DRILL SIZES
ON ANY BASIC SIZE THREAD

for an Approximate 75% Thread

NC/NF INCH & ISO METRIC

Major dia. <u>less</u> thread pitch = Tap drill size

Note: thread pitch = 1.0 inch divided by threads per inch (TPI)

Inch Example:
$(1 \div 16 = .0625)$
3/8 $-$ **16** $= .375 -$ **.0625** $= .3125$ tap drill

Metric Example:
M10 $-$ **1.5** $= 10 -$ **1.5** $=$ M8.5 tap drill

Tool Life Management: In the CURNT COMDS display on a Haas you can PAGE DOWN to the Tool Life Management page. On this page, the Tool Usage register indexes by one every time that tool is called up in the spindle. You enter the number of times you want that tool to be used in the Alarm column. When the Usage number for that tool reaches the number of uses in the Alarm column, it will stop the machine with an alarm. This will help you monitor tools to prevent them from breaking, and prevent parts being scrapped.

STANDARD 60° CENTERDRILL
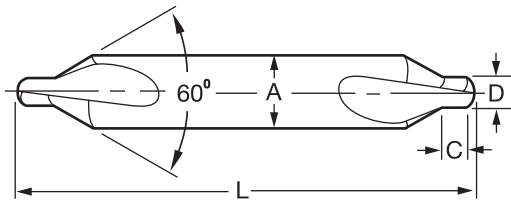
Size	Body Dia (A)	Drill Dia (D)	Drill Length (C)	OAL (L)
00	1/8	0.025	0.030	1 1/8
0	1/8	1/32	0.038	1 1/8
1	1/8	3/64	3/64	1 1/4
2	3/16	5/64	5/64	1 7/8
3	1/4	7/64	7/64	2
4	5/16	1/8	1/8	2 1/8
5	7/16	3/16	3/16	2 3/4
6	1/2	7/32	7/32	3
7	5/8	1/4	1/4	3 1/4
8	3/4	5/16	5/16	3 1/2

Setting 103: CYC START / FH SAME KEY. This is good to use when you're carefully running through a program on a Haas. When this setting is on, the CYCLE START button functions as the Feed Hold key as well. When CYCLE START is pressed and held in, the machine will run through the program; when it's released, the machine will stop in a Feed Hold. This gives you much better control when testing a new program. When you're done using this feature, turn it off. This setting can be changed while running a program. It cannot be on when Setting 104 is on; when one of these settings is turned on, the other will automatically turn off.

To calculate drill tip depth for a chamfer diameter, or drill point depth for a required drilling depth:
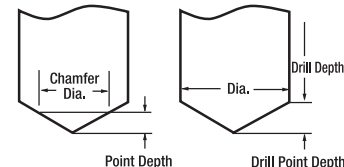
Drill Point Angle (DPA)	Factor
60°	0.866 x Dia. = Point Depth
82°	0.575 x Dia. = Point Depth
90°	0.500 x Dia. = Point Depth
118°	0.300 x Dia. = Point Depth
120°	0.288 x Dia. = Point Depth
135°	0.207 x Dia. = Point Depth

Example: To calculate for a 118-degree drill tip depth, multiply the dia. by 0.3 i.e., 0.250 drill diameter x .3 = 0.075 drill tip depth

Setting 104: JOG HANDL TO SNGL BLK. When Setting 104 is on and a program is running in MEM mode in the Program or Graphics display, pressing the SINGLE BLOCK key allows you to cycle through your program one line at a time, whether the machine is running or you're in Graphics. First press the CYCLE START button, and then each counterclockwise click of the jog handle will step you through the program line by line. Turning the handle clockwise will cause a Feed Hold. This setting can be changed while running a program. It cannot be on when Setting 103 is on; when one of these settings is turned on, the other will automatically turn off.

HAAS

DEGREE FORMULAS

CONVERT MINUTES OF A DEGREE TO A DECIMAL:
Divide minutes by 60
degree minutes to convert: 30° 42'
divide minutes by 60:   42 ÷ 60 = 0.7
bring down degrees: 30.7°

CONVERT MINUTES AND SECONDS TO DECIMAL:
Divide seconds, then minutes by 60
degree minutes and seconds to convert: 30° 41' 15"
divide seconds by 60:   15 ÷ 60 = 0.25
divide decimal minutes by 60:   41.25 ÷ 60 = 0.6875
bring down degrees: 30.6875°

CONVERT A DECIMAL DEGREE TO MINUTES:
Multiply decimal by 60
decimal degree to convert: 30.7°
multiply decimal degree by 60:   0.7 x 60 = 42'
bring down degrees: 30° 42'

CONVERT DECIMAL TO MINUTES AND SECONDS:
Multiply decimal by 60
decimal degree to convert: 30.6875°
multiply the degree decimal by 60: 0.6875 x 60 = 41.25'
multiply decimal minutes by 60:   0.25 x 60 = 15"
bring down degrees: 30° 41' 15"

Editor Quick Cursor Arrow: You can call up a cursor arrow with which to scroll through your program quickly, line by line, when you're in the Editor. For the quick cursor arrow, press F2 once; then you can use the jog handle to scroll line by line through the program. To get out of this quick-cursor mode and remain where you are in the program, just press the UNDO key. (Clasic control only.)

HAAS

HAAS EDIT HOT KEYS (may vary with software version)
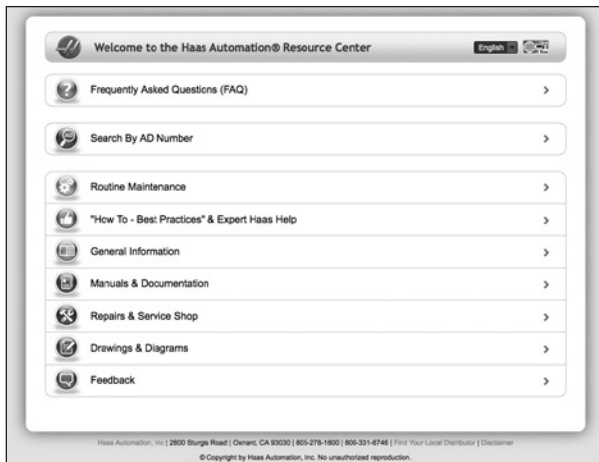
SELECT PROGRAM	When in the EDIT mode, pressing SELECT PROG will bring up the list of programs in the active (highlighted) window.
F2	Press F2 to begin SELECTING A PROGRAM BLOCK to be copied, moved, or deleted. Scroll down to last line of program block. Press either F2 or the WRITE/ENTER key to select block.
INSERT	Press INSERT to copy a selected (highlighted) program block to the line after the one the cursor is on.
ALTER	Press ALTER to move a selected (highlighted) program block to the line after the one the cursor is on.
DELETE	Press DELETE to delete a selected program block that is highlighted.
F4	F4 will paste from clipboard (NGC)
HELP	Displays help information.
F1	Press F1 to access the pop-up menu for easy access to editor functions: HELP, MODIFY, SEARCH, EDIT, and PROGRAM.

Duplicating a Program in LIST PROG: In the LIST PROG mode, you can duplicate an existing program by cursor-selecting the program number you wish to duplicate, typing in a new program number (Onnnnn), and then pressing F2 (on older machines, press F1). On an NGC control, we can duplicate a highlighted program by pressing F3 > Duplicate Program.

SAVING ENERGY MEANS SAVING MONEY

• Auto Power-Off by setting the number of minutes of idle time after which the control will turn itself off

• Power-Off at M30 sets the control to start a 30-second timer that will turn off all power unless interrupted

• Coolant pump shutoff setting

• Screen saver delay setting

• LCD display shutoff setting

• Conveyor shutoff setting

• Servo and hydraulics shutoff setting

greener. innovation.

The Haas Wireless Intuitive Probing System (WIPS) – with optical transmission for part setting, tool setting and inspection – consists of the following elements:

• Spindle Probe Module (work probe)

• Tool Setter Module (tool probe)

• Intuitive Probing System Software (IPS)

• Visual Programming System Software (VPS)

The Haas Wireless Intuitive Probing System makes probing easy to understand and use through simple language, a graphical interface and clear instructions.

Additional information about the Haas Wireless Intuitive Probing System is available from the Haas website (www.HaasCNC.com).

Tool Load Management: Press the PAGE UP or PAGE DOWN key in CURNT COMDS to page to the Tool Load page. Spindle load condition can be defined for a particular tool, and the machine will stop if it reaches the spindle load limit defined for that tool. A tool overload condition can result in one of four actions by the control: ALARM, FEED HOLD, BEEP, or LIMIT FEED. These features are available on the NGC control under the CURNT COMDS > ATM tab.

Leaving Messages: You can enter a message in the MESGS display for the next operator, or for yourself. It will be the first display shown when you power up the machine, if there are no alarms other than the usual 102 SERVOS OFF alarm. If the machine was powered down using EMERGENCY STOP, the MESGS display will not show up when you turn the machine on again. Instead, the control will display the active alarm generated by the emergency stop. In this case, you would have to press the ALARM/MESGS key to view a message. It is not necessary to hit EMERGENCY STOP when you power down a Haas machine.

At Parts.HaasCNC.com, you can find typical service parts and maintenance items, like filters, bulbs, and lubricants – all with up-front pricing. Use your machine serial number to focus on only those parts that will fit. No need to look through hundreds of parts to find the ones that fit your machine – the HaasParts database takes care of it for you.

At DIY.HaasCNC.com, you'll find a library of valuable information – searchable by machine type and topic – like maintenance, how-to videos, manuals, simple repair procedures, and lots more.

Send and Receive Offsets, Settings, Parameters, Macro Variables, Programs, and more to/from Disk. Offsets, settings, parameters, macro variables, ATM information, IPS information, alarm history, keystroke history, linear screw compensation, pallet information, and programs can be saved to a storage device. Press LIST PROG, then select the device to save to or load from. Press F4 and select the appropriate function, then press WRITE.

Send and Receive Offsets, Settings, Parameters, and Macro Variables to/from Disk. For controls using software versions M15.xx and L8.xx and older, offsets, settings, parameters, and macro variables can be saved to or loaded from a storage device. Press LIST PROG, select DESTINATION, and then select an OFSET, SETNG, PARAM, or Macro Variables (PAGE DOWN from CURNT COMDS) display page. Type in a file name, and then press F2 to write to, or F3 to read from disk.

On the Haas Automation YouTube channel, you'll find the most comprehensive collection of instructional and informational videos in the machine tool industry. These valuable tips and tricks are beneficial not only for Haas users, but also all machinists looking to expand their knowledge of CNC machining. New videos are posted weekly, so there's always something great to watch.

Scan to go to
the Haas YouTube
channel now
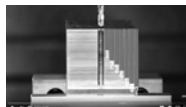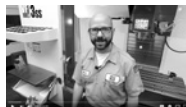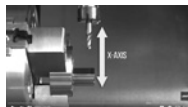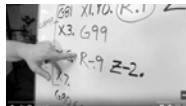
JANUARY

	Mo	Tu	We	Th	Fr	Sa	Su
1	1	2	3	4	5	6	7
2	8	9	10	11	12	13	14
3	15	16	17	18	19	20	21
4	22	23	24	25	26	27	28
5	29	30	31				

FEBRUARY

	Mo	Tu	We	Th	Fr	Sa	Su
5				1	2	3	4
6	5	6	7	8	9	10	11
7	12	13	14	15	16	17	18
8	19	20	21	22	23	24	25
9	26	27	28				

MARCH

	Mo	Tu	We	Th	Fr	Sa	Su
9				1	2	3	4
10	5	6	7	8	9	10	11
11	12	13	14	15	16	17	18
12	19	20	21	22	23	24	25
13	26	27	28	29	30	31	

APRIL

	Mo	Tu	We	Th	Fr	Sa	Su
13							1
14	2	3	4	5	6	7	8
15	9	10	11	12	13	14	15
16	16	17	18	19	20	21	22
17	23	24	25	26	27	28	29
18	30						

MAY

	Mo	Tu	We	Th	Fr	Sa	Su
18		1	2	3	4	5	6
19	7	8	9	10	11	12	13
20	14	15	16	17	18	19	20
21	21	22	23	24	25	26	27
22	28	29	30	31			

JUNE

	Mo	Tu	We	Th	Fr	Sa	Su
22					1	2	3
23	4	5	6	7	8	9	10
24	11	12	13	14	15	16	17
25	18	19	20	21	22	23	24
26	25	26	27	28	29	30	

JULY

	Mo	Tu	We	Th	Fr	Sa	Su
26							1
27	2	3	4	5	6	7	8
28	9	10	11	12	13	14	15
29	16	17	18	19	20	21	22
30	23	24	25	26	27	28	29
31	30	31					

AUGUST

	Mo	Tu	We	Th	Fr	Sa	Su
31			1	2	3	4	5
32	6	7	8	9	10	11	12
33	13	14	15	16	17	18	19
34	20	21	22	23	24	25	26
35	27	28	29	30	31		

SEPTEMBER

	Mo	Tu	We	Th	Fr	Sa	Su
35						1	2
36	3	4	5	6	7	8	9
37	10	11	12	13	14	15	16
38	17	18	19	20	21	22	23
39	24	25	26	27	28	29	30

OCTOBER

	Mo	Tu	We	Th	Fr	Sa	Su
40	1	2	3	4	5	6	7
41	8	9	10	11	12	13	14
42	15	16	17	18	19	20	21
43	22	23	24	25	26	27	28
44	29	30	31				

NOVEMBER

	Mo	Tu	We	Th	Fr	Sa	Su
44				1	2	3	4
45	5	6	7	8	9	10	11
46	12	13	14	15	16	17	18
47	19	20	21	22	23	24	25
48	26	27	28	29	30		

DECEMBER

	Mo	Tu	We	Th	Fr	Sa	Su
48						1	2
49	3	4	5	6	7	8	9
50	10	11	12	13	14	15	16
51	17	18	19	20	21	22	23
52	24	25	26	27	28	29	30
53	31						

2018 HAAS MOTORSPORTS CALENDAR

Series	Race	Date	Track Name
NASCAR	Clash	Sunday, February 11	Daytona International Speedway
NASCAR	Duels	Thursday, February 15	Daytona International Speedway
NASCAR	Daytona 1	Sunday, February 18	Daytona International Speedway
NASCAR	Atlanta	Sunday, February 25	Atlanta Motor Speedway
NASCAR	Las Vegas 1	Sunday, March 4	Las Vegas Motor Speedway
NASCAR	Phoenix	Sunday, March 11	Phoenix International Raceway
NASCAR	Fontana	Sunday, March 18	Auto Club Speedway
NASCAR	Martinsville	Sunday, March 25	Martinsville Speedway
F1	Australia	Sunday, March 25	Melbourne Grand Prix Circuit
NASCAR	Texas 1	Sunday, April 8	Texas Motor Speedway
F1	Bahrain	Sunday, April 8	Bahrain International Circuit
NASCAR	Bristol 1	Sunday, April 15	Bristol Motor Speedway
F1	China	Sunday, April 15	Shanghai International Circuit
NASCAR	Richmond 1	Saturday, April 21	Richmond International Raceway
NASCAR	Talladega 1	Sunday, April 29	Talladega Superspeedway
F1	Azerbaijan	Sunday, April 29	Baku City Circuit
NASCAR	Dover 1	Sunday, May 6	Dover International Speedway
NASCAR	Kansas 1	Saturday, May 12	Kansas Speedway
F1	Spain	Sunday, May 13	Circuit de Barcelona-Catalunya
NASCAR	Charlotte All-Star	Saturday, May 19	Charlotte Motor Speedway
NASCAR	Charlotte Coca-Cola 600	Sunday, May 27	Charlotte Motor Speedway
F1	Monaco	Sunday, May 27	Circuit de Monaco
NASCAR	Pocono 1	Sunday, June 3	Pocono Raceway
NASCAR	Michigan 1	Sunday, June 10	Michigan International Speedway
F1	Canada	Sunday, June 10	Circuit Gilles Villeneuve
NASCAR	Sonoma	Sunday, June 24	Sonoma Raceway
F1	France	Sunday, June 24	Circuit Paul Ricard
NASCAR	Chicago	Sunday, July 1	Chicagoland Speedway
F1	Austria	Sunday, July 1	Red Bull Ring
NASCAR	Daytona 2	Saturday, July 7	Daytona International Speedway

Series	Race	Date	Track Name
F1	Britain	Sunday, July 8	Silverstone Circuit
NASCAR	Kentucky	Saturday, July 14	Kentucky Speedway
NASCAR	New Hampshire	Sunday, July 22	New Hampshire Motor Speedway
F1	German	Sunday, July 22	Hockenheimring
NASCAR	Pocono 2	Sunday, July 29	Pocono Raceway
F1	Hungary	Sunday, July 29	Hungaroring
NASCAR	Watkins Glen	Sunday, August 5	Watkins Glen International Raceway
NASCAR	Michigan	Sunday, August 12	Michigan International Speedway
NASCAR	Bristol 2	Saturday, August 18	Bristol Motor Speedway
F1	Belgian	Sunday, August 26	Circuit de Spa-Francorchamps
NASCAR	Darlington	Sunday, September 2	Darlington Raceway
F1	Italy	Sunday, September 2	Autodromo Nazionale Monza
NASCAR	Indianapolis	Sunday, September 9	Indianapolis Motor Speedway
NASCAR	Las Vegas 2	Sunday, September 16	Las Vegas Motor Speedway
F1	Singapore	Sunday, September 16	Marina Bay Street Circuit
NASCAR	Richmond 2	Saturday, September 22	Richmond International Raceway
NASCAR	Charlotte 2	Sunday, September 30	Charlotte Motor Speedway
F1	Russia	Sunday, September 30	Sochi Autodrom
NASCAR	Dover 2	Sunday, October 7	Dover International Speedway
F1	Japan	Sunday, October 7	Suzuka International Racing Course
NASCAR	Talladega 2	Sunday, October 14	Talladega Superspeedway
NASCAR	Kansas 2	Sunday, October 21	Kansas Speedway
F1	United States	Sunday, October 21	Circuit of the Americas
NASCAR	Martinsville 2	Sunday, October 28	Martinsville Speedway
F1	Mexico	Sunday, October 28	Autodromo Hermanos Rodriguez
NASCAR	Texas 2	Sunday, November 4	Texas Motor Speedway
NASCAR	Phoenix 2	Sunday, November 11	Phoenix International Raceway
F1	Brazil	Sunday, November 11	Autodromo Jose Carlos Pace
NASCAR	Homestead-Miami	Sunday, November 18	Homestead-Miami Speedway
F1	Abu Dhabi	Sunday, November 25	Yas Marina Circuit

HAAS

_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____
_____

Haas Automation, Inc. USA
2800 Sturgis Road, Oxnard, CA 93030
P. 800-331-6746 I www.HaasCNC.com

Haas Automation, Europe
Mercuriusstraat 28, B-1930
Zaventem, Belgium
P. ++32-2-522 99 05 I www.HaasCNC.com

HAAS

WHAT'S INSIDE THIS BOOKLET?

Decimal Equivalent Chart / Millimeter to Inch Chart
Haas Mill G-Codes / Haas Mill M-Codes
Haas Lathe G-Codes / Haas Lathe M-Codes
Abbreviations and Measurement Units
Mill and Lathe Formulas
Tapping and Threading Formulas
Tap Drill Calculation
Drill Point Depth & Countersink Formulas
Degree Formulas

Haas Automation, Inc. USA
2800 Sturgis Road, Oxnard, California 93030
Toll Free: 800-331-6746 | www.HaasCNC.com

Haas Automation, Europe
Mercuriusstraat 28, B-1930 Zaventem, Belgium
Tel: +32 2 522 99 05

www.HaasCNC.com