

Contents

Introduction	1
G Code List	2
M Code List	5
Sample Part	7
Sample Program	8
Tool Layout	17
How To Take Tool Offsets	18
Notes on taking offsets:	21



Introduction

The Purpose of this document is to give additional information for the instructional videos. Please click on the Table of Content to jump to the desired information.

G Code List

G code			FUNCTION EXPLANATION
Type A	Type B	Type C	
G00	G00	G00	Positioning (Rapid traverse)
G01	G01	G01	Linear interpolation (Cutting feed)
G02	G02	G02	Circular interpolation CW
G03	G03	G03	Circular interpolation CCW
G04	G04	G04	Dwell
G04.1	G04.1	G04.1	Waiting (Main and sub spindle program execute)
G17	G17	G17	XY plane selection
G18	G18	G18	ZX plane selection
G19	G19	G19	YZ plane selection
G90	G77	G20	Outer diameter / internal diameter cutting cycle
G92	G78	G21	Thread cutting canned cycle X axis
G21.1	G21.1	G21.1	Thread cutting canned cycle Y axis
G94	G79	G24	Face turning canned cycle
G28	G28	G28	Reference point return
G29	G29	G29	Return from reference point
G30	G30	G30	2nd, 3rd, 4th reference point return
G32	G33	G33	Thread cutting
G40	G40	G40	Tool nose radius compensation cancel
G41	G41	G41	Tool nose radius compensation left
G42	G42	G42	Tool nose radius compensation right
G43	G43	G43	Tool length compensation (+)

G44	G44	G44	Tool length compensation (-)
G49	G49	G49	Tool length compensation cancel
G52	G52	G52	Local coordinate system setting
G53	G53	G53	Machine coordinate system setting
G54~G59	G54~G59	G54~G59	Work coordinate system setting
G65	G65	G65	Macro call
G66	G66	G66	Macro Modal call
G67	G67	G67	Macro modal call cancel
G20	G20	G70	INCH command
G21	G21	G71	METRIC command
G70	G70	G72	Finishing cycle
G71	G71	G73	Longitudinal turning cycle
G72	G72	G74	Face turning cycle
G73	G73	G75	Pattern repeating
G74	G74	G76	End face peck drilling (Z axis)
G75	G75	G77	Outer diameter / internal diameter drilling cycle (X、 Y axis)
G77.2	G77.2	G77.2	Outer diameter / internal diameter drilling cycle (Y axis)
G76	G76	G78	Multiple thread cutting cycle X axis
G78.2	G78.2	G78.2	Multiple thread cutting cycle Y axis
G80	G80	G80	Drilling canned cycle cancel
G83	G83	G83	Drilling canned cycle (Z axis)
G84	G84	G84	Tapping canned cycle (Z axis)
G85	G85	G85	Boring canned cycle (Z axis)
G87	G87	G87	Longitudinal drilling canned cycle (X axis)

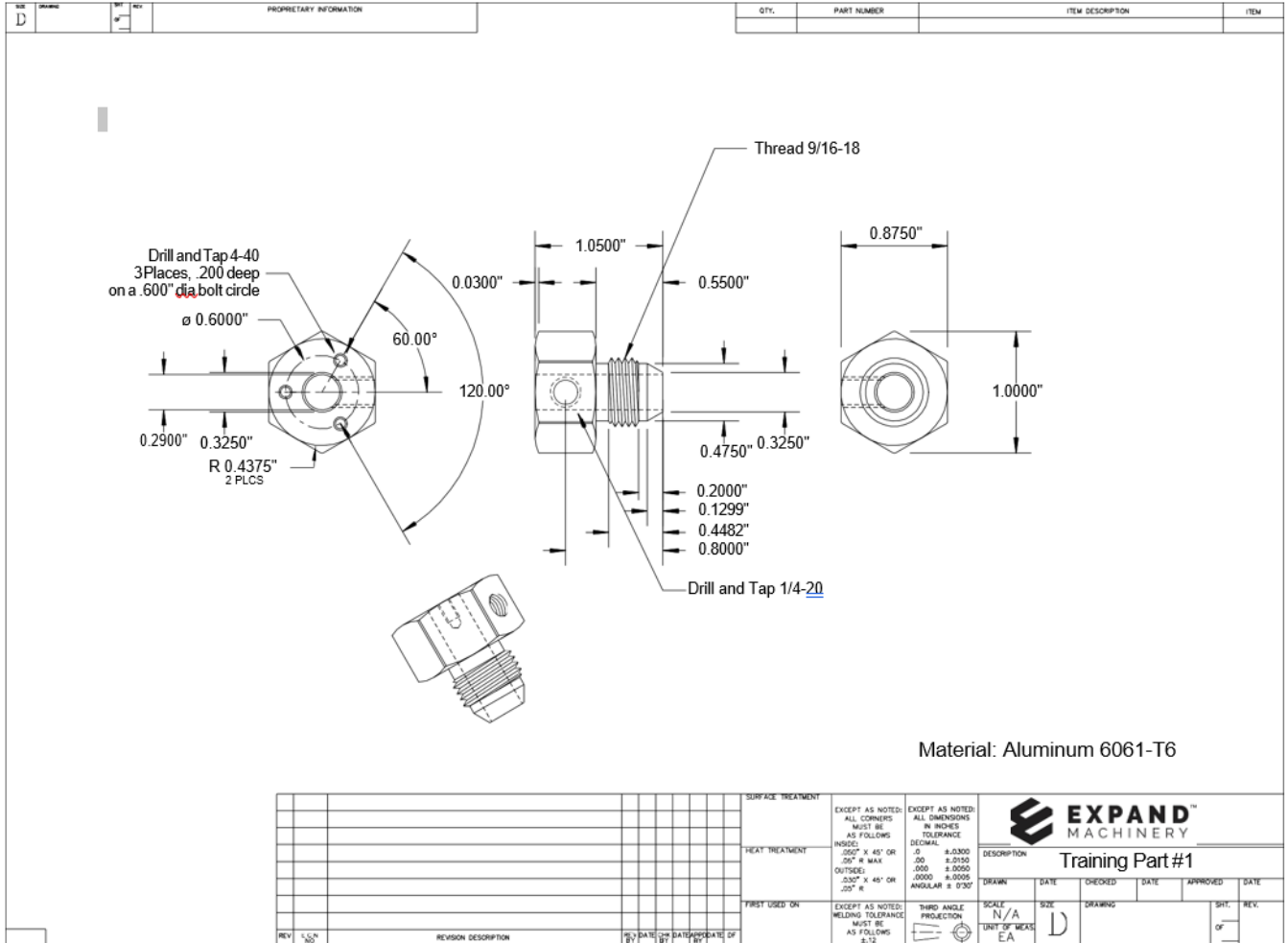
G88	G88	G88	Longitudinal tapping canned cycle (X axis)
G89	G89	G89	Longitudinal boring canned cycle (X axis)
---	G90	G90	Absolute programming
---	G91	G91	Incremental programming
G50	G92	G92	Coordinate system setting or max spindle speed setting
G98	G94	G94	Feed per minute (mm/min)
G99	G95	G95	Feed per revolution (mm/rev)
G96	G96	G96	Constant surface speed control (m/min)
G97	G97	G97	Constant surface speed control cancel
---	G98	G98	Return to initial level
---	G99	G99	Return to R point level

M Code List

M Code	Function Description	
	Main spindle program	Sub spindle program
M00	Program Stop	
M01	Optional Program Stop	
M03	Main Spindle CW (Rigid tapping)	Sub Spindle CW (Rigid tapping)
M04	Main Spindle CCW (Rigid tapping)	Sub Spindle CCW (Rigid tapping)
M05	Main Spindle Stop (Rigid tapping)	Sub Spindle Stop (Rigid tapping)
M08	Coolant ON	
M09	Coolant OFF	
M10	Main Spindle Chuck Close	Sub Spindle Chuck Close
M11	Main Spindle Chuck Open	Sub Spindle Chuck Open
M16	Parts Catcher Forward	
M17	Parts Catcher Backward	
M19	Main Spindle Switch to C Axis	Sub Spindle Switch to C Axis
M20	Main Spindle Switch to C Axis Cancel	Sub Spindle Switch to C Axis Cancel
M28	Coolant 2 ON	
M29	Coolant 2 OFF	
M30	Program End	
M31	Chuck Open Allowed When Main Spindle Rotating (For Bar Change)	Chuck Open Allowed When Sub Spindle Rotating (For Bar Change)
M32	Chuck Open Allowed When Main Spindle Rotating (For Bar Change) Cancel	Chuck Open Allowed When Sub Spindle Rotating (For Bar Change) Cancel
M37	Chip Conveyor ON	
M38	Chip Conveyor OFF	
M Code	Function Description	
M39	Main Spindle C Axis Positioning	Sub Spindle C Axis Positioning

M40	Main Spindle C Axis Brake ON	Sub Spindle C Axis Brake ON
M41	Main Spindle C Axis Brake OFF	Sub Spindle C Axis Brake OFF
M55	Chuck ON (Have Material)	
M56	Chuck OFF (Have Material)	
M57	Main Spindle Air-Blowing ON	Sub Spindle Air-Blowing ON
M58	Main Spindle Air-Blowing OFF	Sub Spindle Air-Blowing OFF
M59	Sub Spindle Part Ejector with Air Blowing ON	
M60	Sub Spindle Part Ejector with Air Blowing OFF	
M66	After spindle rpm reached and next command Start	
M69	Sub Spindle Part ejector ON	
M70	Sub Spindle Part ejector OFF	
M71	Command Main Spindle As Machining Spindle	Command Sub Spindle As Machining Spindle
M72	Command A Axis As Machining Spindle	Command A Axis As Machining Spindle
M73	Command B Axis As Machining Spindle	Command B Axis As Machining Spindle
M74	Command Sub Spindle As Machining Spindle	Command Sub Spindle As Machining Spindle
M75	Command B2 Axis As Machining Spindle	Command B2 Axis As Machining Spindle
M77	Main Spindle and Sub Spindle Simultaneous ON	
M78	Main Spindle and Sub Spindle Simultaneous OFF	
M79	Z1 and Z2 Simultaneous ON	
M80	Z1 and Z2 Simultaneous OFF	
M81	Bar Feeder Thrust Pause	
M82	Cancel Bar Feeder Thrust Pause	
M83	Block Skip ON	
M84	Block Skip OFF	
M98	Call Sub-Program	
M99	Sub-Program Return to Main Program	

Sample Part



Sample Program

\$1

(Mat Hex 7/8 6061-T6)
(Part Out 2.5")
(T1-Turning / Facing Holder-PCLCR 08-3S Insert-CCMT 3-1-SM)
(T2-Thread Profile Holder-PVACR 08-2S Insert-VCMT 220-5M)
(T3-Threading Tool 9/16-18 Holder-SCHR 12.7-6B Insert-SCIR 6B-MTR007)
(T28- 3/8" Flat EM)
(T11-Spot ¼")
(T12-Drill #7)
(T14-Tap ¼-20)
(T6-Part Off Holder-DGTL 12.7B-2D30 Insert-DGN2202J)
(T31-Spot ¼")
(T33-Drill #43 .089" Dia.)
(T34-Tap 4-40)

N1001
G4.1P1
G53Z0
G50S4000

N1 (TURNING FACING)
G53Z-2. (Safe Index)
T101
G97S1=4000M103
G0G99X1.1Y0
Z.1
G71U.05R.02
G71P100Q200U.01W.01F.01
N100G0X.325
G1Z0F.003
X.475Z-.1299
X.559
Z-.545
X.875
X1.Z-.575
N200X1.1

pg. 8

818 349-9166

www.expandmachinery.com/training

G70P100Q200
G53Z-2. (Safe Index)
M1

N2 (PROFILE TOOL)
G53Z-2. (Safe Index)
T202
G96S1=600M103
G0G99X.6625Y0
Z-.4482
G1X.4750F.005
Z-.55
X.875
X1.1Z-.580F.002
G53Z-2. (Safe Index)
M1

N3(THREADING OD 9/16-18)
G53Z-2. (Safe Index)
T303
G97S1=400M103
G0G99X.6Y0
Z.1
G76P010060Q.002R.002
G76X.4943Z-.525P.0341Q.015F.055555
G53Z-2. (Safe Index)
M105
M1

N21 (SPOT DRILL)
G53Z-2. (Safe Index)
T2121
G97S1=4000M103
G0G99X0Y0
Z.1
G1Z-.097F.01
G53Z-2. (Safe Index)
M1

N22 (DRILL #L)
G53Z-2. (Safe Index)
T2222
G97S1=2500M103
G0G99X0Y0
Z.1

G83Z-1.59Q.05F.01
G53Z-2. (Safe Index)
M1

N28
G53Z-2. (Safe Index)
T2828

G53Z-2. (Safe Index)

N11(SPOT DRILL SIDE LIVE)
G53Z-2. (Safe Index)
T1111
G97S2=4000M203
G0G98X1.1Y0
M19
G28C0
G0C0
Z-.8
G1X.750F5.
G0X1.2
G99
G53Z-2. (Safe Index)
M1

N12 (DRILL #7)
G53Z-2. (Safe Index)
T1212
G97S2=4000M203
G0G98X1.1Y0
M19
G28C0
G0C0
Z-.8
G87X0Q.05F5.
G80
G0X1.2
G99
M205
G53Z-2. (Safe Index)
M1

N28(3/8 EM Axial Live SIDE LIVE)

G53Z-2. (Safe Index)
T2828
G97S3=4000M303
G0G98X1.5Y0
M19
G28C0
G0C0
Z.1

(G12.1 Hex Training Part)

G12.1
G0 X1.254 C1.024
G1Z-.75F30.
C-.724F16.5
X0C-1.448
X-1.254C-.724
C.724
X0C1.448
X1.5138C.574
G0Z.1
X1.254C1.024
Z-.65
G1Z-1.1F10.8
C-.724F16.5
X0C-1.448
X-1.254C-.724
C.724
X0C1.448
X1.5138C.574
G0Z.1
X1.25C1.0217
G1Z-1.1F10.8
C-.7217F65.4
X0C-1.4434
X-1.25C-.7217
C.7217
X0C1.4434
X1.5098C.5717
G0Z.1
G13.1

G99
G53Z-2. (Safe Index)
M1

N14 (1/4-20 TAP)
G53Z-2. (Safe Index)
T1414
G97S2=500M203
G0G99X1.1Y0
M19
G28C0
G0C0
Z-.8M72
G88X0F.05
G80
G0X1.2
M71
G53Z-2. (Safe Index)
M1

M204S2=4000
G4X2.
M205

N800

N1002
G4.1P2
G53Z-2. (Safe Index)

N6 (PART OFF)
T606
G0Y0
Z-1.146 (PRESENT PART TO SUB)

N1003
G4.1P3
(Wait for sub to grab part and sync)
N1004
G4.1P4

M11
G4X.5
W1.156
M10
G4X.5
M79

T606
G97S1=1500M103
G0X1.1
G1X-.03F.002
G0X1.1
W.04
G1X1.F.03
X.875W-.03F.002
X-.03
G0X1.1
M78 (un-sync sub rotation)
M80 (un-sync Z1-Z2)
N1005
G4.1P5

M100 (Check Part Off)
G53Z-2. (Safe Index)

G4.1P1000
(BAR CHANGE)
M83
(/M98P9999)
M84
G4.1P1001
M99

\$2
N1001
G4.1P1
G53Z0 (Safe Index)

N31
G53Z0 (Safe Index)
T3131(SPOT DRILL)
G97S5=4000M503
G0G98X.6M28
M19
G28C0
G0C0
C60.Z.1
G83Z-.0479Q.150F5.
C180.Q.150
C300.Q.150

X0Z-.1625Q.2625
G80
G53Z0 (Safe Index)
M1

N33
G53Z0 (Safe Index)
T3333(#43 DRILL)
G97S5=4000M503
G0G98X.6M28
M19
G28C0
G0C0
C60.Z.1
G83Z-.380Q.05F5.
C180.Q.050
C300.Q.050
G80
G53Z0 (Safe Index)
M1

N34
G53Z0 (Safe Index)
T3434(4-40 TAP)
G97S5=500M503
G0G99X-.6M28 (X minus for tool clearance)
M19
G28C0
G0C0
M75
C240.Z.1
G84Z-.350F.025
C0
C120.
G80
M74
M504
G4X2.
M505
G53Z0 (Safe Index)
M1

(Part Eject)
N35
G53Z0 (Safe Index)
T3535 (part catcher)
G97S4=150M403
G0Z0
X0
M31
M16 (part catcher forward)
M11 (open collet)
G4X.5 (dwell)
M69 (part ejector forward)
G4X.5 (dwell)
M70 (part ejector back)
M57 (air blow on)
G4X1. (Dwell)
M58 (air blow off)
M17 (part catcher back)
G4X.5 (dwell)
M405
M32
G53Z0 (Safe Index)
G53X0
M1

N800

N1002
G4.1P2

G53Z0
G53X0
M11

N1003
G4.1P3

N30
T3030

M57
G0X0
Z.2
G98G1Z-.950F30. (How far sub spindle onto part)

M58 (air blow off)
M10 (chuck close)
G4X.2 (dwell)
M77 (sync sub rotation)
N1004
G04.1P4
(Wait for part off)
N1005
G04.1P5
M78 (un-sync sub rotation)
M80 (un-sync Z1-Z2)
G99 (CHANGE BACK TO IPR)
M100
G53Z0 (Safe Index)
M405 (spindle off)
G4.1P1000
(Bar change)
G04.1P1001
M99

How To Take Tool Offsets

The offsets on the 32CS are the distance from machine zero to the part zero for each tool in each axis. Since the machine does not have a turret it has to use the offsets to determine where each tool is in relation to the part.

To move a tool to the part we simply activate the offsets using a “T” command and call out a position we would like to the tool to move.

Caution!!! Be aware that moving a tool in position may cause another tool to move into the part. It is always a safe practice to use the Z axis to move the part out of the way before using MDI to position the tool in the X and Y axis.

Some offsets are set by the factory and will remain constant unless the machine zero or tooling plate is changed.

It is highly recommended that these offsets be recorded so they can be restored if lost or changed.

These offsets are:

Tools #1 - #6	Y
Tools #11 - #14	Y
Tools #21 - #28	X&Y
Tools #31 - #34	X2

Tools #1 thru #6 Turning Tool Offsets:

Tools 1 - 6 “Y” axis offset should be set as a constant with a new machine. Tools 1-6 “Y” offset will change if the tooling plate is changed from 1/2” to 5/8” shank tooling.

How To Call up the Tools 1-6 using MDI

1. Manually move the Z axis to a position where all tools will clear the material.
2. In MDI execute the command “G0 T101 Y0”. This will bring tool #1 into position to make a turning cut.

To set the X offset Tools 1 - 6

1. Turn the spindle on in MDI and use X and Z in manual operation to turn a diameter on the material in the spindle. In MDI execute the command “M103 S1=1000”. This will turn the spindle on at 1000 RPM.
2. Move the tool to a position in Z away from the part.
3. Press the “POS” button to go to the position page.
4. Press “F2” Clear X Relative (Note that the “Relative Register” X value is “0”).
5. Use the hand wheel to move the X axis to 7.000”. This will give you enough room to measure the part.

6. Measure the diameter of the cut (Example: .995”).
7. Press “MON” to go to the Monitor page.
8. Press “F5 - Tool Setting” to go to the tool offset page.
9. Press “F2” “Tool Len”
10. Place cursor on the tool offset #1.
11. Press “X7.995” “Enter”
12. The correct offset for the tool should be entered in the offset value.

To set the Z offset Tools 1 - 6

1. Call up the tool (shown above).
2. Move the tool to a known position in relation to the Z0 (face of part) using a gauge pin/block or just use the tool to cut a face. (Example: touch face of part with .250” gauge pin.
3. Press monitor
4. Press “F5” Tool Setting (Under Screen)
5. Press “F2” “Tool Len”
6. Place cursor on the tool #1 offset.
7. Press “Z.250” “Enter” (enter Z0 if you have just cut the face)
8. The correct offset for the tool should be entered in the offset value.

Tools #11 thru #14 Radial Driven Tool Offsets:

Tools 11-14 are radial driven tools used to do Milling type work on the side of the part. The “Y” axis on these tools will be a constant. It is recommended to write down these values and keep them in a safe place in case they are lost.

How To Call up the Tools 1-6 using MDI

1. Manually move the Z axis to a position where all tools will clear the material.
2. In MDI execute the command “G0 T1111 Y0”. This will bring tool #11 into position to make a turning cut.

How To set the X offset Tools 1 - 6

1. Turn the spindle on in MDI and use X and Z in manual operation to touch or create a diameter on the material in the spindle. In MDI execute the command “M303 S3=1000”. This will turn the spindle on at 1000 RPM.
2. Move the tool to a position in Z away from the part.
3. Press the “POS” button to go to the position page.
4. Press “F2” Clear X Relative (Note that the “Relative Register” X value is “0”).
5. Use the hand wheel to move the X axis to 7.000”. This will give you enough room to measure the part.
6. Measure the diameter of the cut (Example: .995”).
7. Press “MON” to go to the Monitor page.
8. Press “F5 - Tool Setting” to go to the tool offset page.
9. Press “F2” “Tool Len”
10. Place cursor on the desired tool offset.

11. Press "X7.995" "Enter".
12. The correct offset for the tool should be entered in the offset value.

How To set the Z offset Tools 1 - 6

1. Call up the tool (shown above).
2. Move the tool to a known position in relation to the Z0 (face of part) using a gauge pin/block or just use the tool to cut a face. (Example: touch face of part with .250" gauge pin.
3. Press monitor.
4. Press "F5" Tool Setting (Under Screen)
5. Press "F2" "Tool Len"
6. Place cursor on the tool #1 offset.
7. Add the gauge pin to the radius of the tool. Example 3/8" End Mill radius=.1875 (Add .25 + .1875=.4375)
8. Press "Z.4375" "Enter" (enter "Z.1875" if you have just cut the face with the tool)
9. The correct offset for the tool should be entered in the offset value.

Tools #21 (Fixed) thru #28 (Axial Driven) Tool Offsets:

How To Set the X and Z offset Tools 21-25

Tools 21-25 are fixed bushing holders. They may be used for Drilling or Boring operations.

Drilling

When using Tools 21-25 for drilling operations the Z axis is the only axis offset you need to get.

You can use the same procedure used on tools 1-6.

Boring

When you use tools 21-25 in a boring operation you can set the X and Z offsets using the same procedure as tools 1-6.

Notes on taking offsets:

When you enter an axis letter in the “offset input line” Example “X.995” the offset will be calculated using the value in the “Machine Register”.

When you enter a number in the “offset input line” without an axis letter Example “.995” the value will be input to the offset selected according to the setting of the “INPUT MODE” in the upper left corner of the screen. To change this mode input “I” for incremental input or “A” for absolute input of the number. The active mode will be displayed to the right.

(Absolute mode shown below)

Input Mode (A:ABS I:INC) ABS

Absolute Input Mode will enter the number into the offset.

Incremental Input Mode will add the number to the existing offset. (If a negative value the sign must be included)

Special Tool Numbers T3030 and T3535

Tool Offset #30 is used for the “Face of the sub spindle collet” to the “Face of the part when Z1 is at the transfer position”.

Tool Offset #35 is used for the position of the sub spindle over the part catcher when extended.