

A20-Type VII (A220, Fanuc controller)

Note: Up to 3 M-codes can be commanded in a single line. It saves time to command as many M-codes as possible in one line. Some M-codes are macros; only a single macro M-code can be in a line.

See (macro) below.

Machining Data

8570=1

(#8570=1 ALLOWS CONTROLLER TO READ INTERNAL CODES)

(The following are read from MC Data)

(#15200=Material Diameter)

(#15201=Position Point)

(#15202=Machine Length)

(#15203=Pieces per Chuck)

(#15204=Cutoff Tool Number)

(#15205=Cut-Off Spindle Speed)

(#15206=Cut-Off Feed rate)

(#15207=Cut-Off End Point)

(#15208=Back Work Piece Extend Length, aka back spindle chuck position)

(#15209=Front Machining Holder)

(#15211=Back Machining Tool Holder)

(#15212=Back Spindle)

(#15215=Back long neck chuck extend length, aka length of extended nose on pick off collet)

#8570=0

(#8570=0 locks access to internal codes)

M00- Program stop. Force the machine to stop while executing a program. Pushing cycle start will resume operation. Citizen machines also stop spindles

M01- Optional stop

M1- If the offset OPR page OPTIONAL STOP button is on, the executing line \$1 or \$2 will stop. Pushing CYCLE START will resume operation. If the OPTIONAL STOP button is off, the M01 command is skipped.

M02- (macro) Main program one cycle stop.

M2- M2 will queue lines \$1-\$2 then stop execution of the program if in ONE CYCLE MODE. Put right before M99 of main program for both \$1 and \$2. M2 is how the controller knows a program is a "Main" program and not a "Sub" program.

M03- (macro)

M3- Front spindle forward rotation. Format is "M3 S1=1000" not "M3S1000", 8k RPM max, 200 RPM min. No other codes are allowed on the programming line. Other machines often use "M3 S1000", sometimes a spindle designator is required so the code may look like "M3S1000P1".

M04- (macro)

M4- Front spindle reverse rotation. Same format at M3

M05-

M5- Main spindle stop

M06- (macro)

M6- Collet close (front spindle). Use it alone on the line!

M6

M3S1=1000(Will work)

M6U.5 (U argument added to dwell after chuck closed)

M07- (macro)

M7- Front spindle collet open. Must be use alone on a line. M7 will not open if the spindle is rotating over 800RPM, use M700 to open the collet at any spindle speed.

M7U.5 (U argument added to dwell after chuck opened for bar feeders)

M08- M8- Short cut program. Used for bar loader sub-program. M8 shifts control of the BLOCK SKIP function to the bar loader. This lets one section of a program or a sub-program run only if the end-of-bar switch on the bar feeder is tripped.

Main Program

M9(beginning- safety)

"

"

G1X-.1F.002(CUT-OFF)

M8

M8(CONFIRM M8)

/M98P8000(if EOB) The "/" is the BLOCK SKIP function. If the BLOCK SKIP button is activated the line is skipped over.

M9

M09- (macro)

M9- Cancels M8. Returns control of the BLOCK SKIP function to the BLOCK SKIP button. If using M8 then put a M9 at the beginning of your program as a safety command if starting the program after a problem.

M10- Part knock out rod advance. If the collet is closed or the spindle is running the rod will not advance and you will get an error. During operation two conditions can cause an error: The knock out rod does not make it all the way to its stroke end position or it encounters something that causes a spring loaded mechanism to trip. The speed of the rod can be adjusted by a opening or closing an air valve.

M11- Part knock out retract.

M15- (macro) Back spindle collet close. Use it alone on the line! M15U.5 (U argument added to dwell after chuck closed)

M16- (macro) Back spindle collet open. Same operation rules as M15

M18- (macro) (only with "C" axis option) References the "C" axis and goes to position specified. Only used if simple indexing in 1 deg. increments and to enable the axis. To feed "C" see below.

M18 C0

G1 XYZ

M18 C180. (M18 issued a G0 C?? internally)

XYZ

-If we want to use "C" axis and rotate it then

M18 C0

G98G1C150.F1000. (feed=deg. per min) (Feed command is F360. = 360DPM)

H120. (incremental)

-If you need to change the zero pos for M28 S0 to mach a hex flat use parameter \$1 #4135. 360deg = 360000 so 1 deg = 1000

-To save the most cycle time do not use M5

M3S1=4000

"

"

M18 C15. (No M5 needed).

M20- (macro) Cancel "C" axis mode. M3-M4-M5 will also do the same.

M23- (macro) Back spindle forward rotation "M23 S2=1000", 8000 RPM max, 200 RPM min. See M3 for other brand machines format.

M24- (macro) Back spindle reverse rotation "M24 S2=500"

M25- (macro) Back spindle stop

M28- (macro) (Non "C" axis machine only) Spindle indexing.

1 degree increments only. Only absolute commands.

M28 S15

M28 S0

M28 S180

-If you need to change the zero pos for M28 S0 to mach a hex flat use parameter \$1 #4135. 360deg = 360000 so 1 deg = 1000 (Cincom manual shows the wrong parameter).

-To save the most cycle time do not use M5

M3S1=4000

"

"

M28 S0 (no M5 needed)

M31- Work conveyor on. (option) This is automatically called from M34.
Timer = T24 12000 = 12sec 'Parameter'-'PMCMNT'-'TIMER'

M33- (macro) Parts ejection from back spindle without knock-out. This is similar to M34 but doesn't dump the part into the tray. This is left for you to do your own unique way. M33 will move Z2 to home position then to the parts dump position. Now you can program how you want the part to dump. M33 can only be commanded from \$2. The only reason to use M33 is to save cycle time, or to have a special ejection routine.

M33 W0 D250. X.156 E400.

M33 -Nothing else on the line then move as explained above.

W -After the part is caught then the catcher moves back to Z0. or the amount of W from Z0. Normally not used.

D -Feed rate for Z2 to move away from GB.

U -Used to change the position of X2 at the dump position of ".156". Normally don't use.

E -Feed rate for X2 to move to the dump position.

\$2

M33 (position to dump)

G0Z1.299 (33mm move into parts chute)

M16 (open PO chuck)

M10 (knock out)

G4U.2 (confirm M10)

Z0M11

M34- (macro) Series of commands needed for parts ejection from the back spindle all in one command. Collet open, parts catcher in/out, knock out forward, air etc. This can be commanded from \$1 or \$2 but has two very different functions. If M34 in \$1 then dump the part from the catcher mounted on the back spindle. If M34 from \$2 then eject the part from the back spindle into the parts box. There are many command options that can be used but usually just "M34" is all you need. See the different command options below.

If M34 from \$2 and part is in the sub spindle then...

M34 W D U E Z F B T S X

M34 -nothing else on line then catcher moves away and ejects the part.

W -After the part is caught then the catcher moves back to Z0 or the amount of W from Z0. Normally not used.

D -Feed rate for Z2 to move away from GB. IPM

U -Used to change the position of X2 at the dump position of ".156". Normally don't use.

E -Feed rate for X2 to move to the dump position. IPM

Z -Move to Z position inside collection box.

F -Feed rate when moving into position inside collection box. IPM

B -When finished move Z2 to amount B from Z0.

T -Timer for air blow after part is ejected.

S -Back Spindle rotation speed when using optional basket unloading.

X -X2 position to go to after unloading the part.

M48- (macro) Back spindle "C" axis on. (only with "C2" axis option) References the "C" axis on the back spindle and goes to position specified. Only used if simple indexing in 1 deg increments and to enable the axis.

M48 C0

G1 XYZ

M48 C180. (M48 issued a G0 C?? internally)

XYZ

-If we want to use "C" axis and rotate it then

M48 C0

G98G1C150.F1000. (feed=deg. per min) (Feed command is F360. = 360DPM)

H120. (incremental)

-If you need to change the zero pos for M48 S0 to mach a hex flat use parameter \$2 #4135. 360deg = 360000 so 1 deg = 1000

-To save the most cycle time do not use M25

M23S2=4000

"

"

M48 C15. (no M25 needed)

M50- Check cut-off tool breakage by spindle RPM. You command the main spindle and then use M50. M50 will check if S2 is turning, if so then alarm.

G814

M3S1=2000

G1X-.1(cut part off of bar)

M5 (stop front spindle)

M25 (stop back spindle)

G113 (turn off spindle synch.)

M3S1=100 (rotate front spindle at 100 rpm)

G4U.5

M50 (if S2 is turning then alarm)

G600

"

M51- (macro)

Tool breakage detection, by using a switch. After cut-off move Z back a little then position over the switch. This will check if the part is gone. If not then Alarm. The switch is left of T100.

M51 X.3 W-.2 F.002

X= The X position to move to during checking. If no X is used then X-1.mm or X-.04 is default.

W= The incremental amount to move back so the switch contacts the proper spot of the part and away from the cut-off tool. If no W is used then Z stays where it is for the check.

F= IPM feed rate to check. If no F then 2000mmpm or 78.7IPM is default.

\$1

G50Z-.005

M6

G0X.8Z-.02

M51 W-.02

T200 (continue on)

Or

\$1
G50Z-.005
M6
G0 X.8 W-.02

M51 (use all default settings)
T200 (continue on)

- M52- Coolant oil ON
- M53- Coolant oil OFF
- M54- Bar loader feeding pressure stop -BAR LOADER ONLY
- M55- Bar loader material replace -Tells bar loader to load a new bar. Then waits until the new bar is in position. See M8.
- M56- Counter- Increments parts counter by one if in CONTINUOUS CYCLE MODE
This can be changed to any CYCLE MODE by changing PC-PRM KEEP-RL # K00 bit 4 to a 1.
- M61-
M62-
M63- EXTRA M-CODES FOR SPECIAL CUSTOMER USE
M64-
M65-
B-
B1- B1 to B255 are also available if using a PLC to read the signals. With M61-M65 there are 5 possible functions. B has 256!
- M72- Back spindle air blow ON -clears chips from back spindle collet. If a long workpiece gripper option is used, M72 closes the gripper.
- M73- Back spindle air blow OFF. If a long workpiece gripper option is used, M73 opens the gripper.
- M77- Wait for spindle Synch -See G814. It is best to use M77 in \$2 just before the M15 (close the back spindle collet) but M77 can be used anywhere after G814.
- M78- (macro) Currently an option, use M48 as C is standard. Back spindle index and lock - "M78S0" S=0 to 360 in 1 deg. increments. This is absolute indexing. Incremental indexing can be done with macro commands. Also "M78 S180" must be on one line in the program alone
- M79- Unlock back spindle indexing.
- M80- (macro) Gang Tool live spindle forward revolution. "M80 S2000" You need to have M80/81 and S#### on one line. 6000 RPM max 200 min., 8000 RPM is possible but over suggested rating for long cycle time of operation) M80S3000 or M80S3=3000
- M81- (macro) Gang Tool live spindle reverse revolution, M81S3000 or M81S3=3000
- M82- (macro) Gang Tool live spindle stop
- M86- Cutting start interlock enabled. Controller will wait for the spindle to get to speed before continuing.
- M87- Cutting start interlock disabled. Controller will not wait for the spindle to get to speed before continuing
- M88- Interference check disabled. Be careful!
- M89- Interference check enabled. Good to include as safety code at start of program.
- M92- Error detect On- Modal exact program stop. Used so corners are not rounded off while turning at high feed rates. This gives small dwells to confirm corners. See also G9
- M93- Error detect off. Exact stop cancel. Cancels M92
- M94- Back Spindle speed fluctuation detect on (same as G26)

- M95- Back Spindle speed fluctuation detect off (same as G25), used during rigid tap or constant SFM
- M96- Main Spindle speed fluctuation detect on (same as G26)
- M97- Main Spindle speed fluctuation detect off (same as G25), used during rigid tap or constant SFM
- M98- Sub program call. If commanded "M98 P1111 L5" from line \$1 of a main program then the line \$1 of sub program O1111 is repeated 5 times. When the sub program is finished M99 returns to the line after the M98 command in the main program. Make sure your sub program has a \$2 and M99 even if you are not using \$2.
- M99- Rewind: If commanded in sub program then return from sub program to main program. If commanded in the main program, then return to the beginning. Also M99 can use a "P" argument. "M99 P10" this would return to line "N10", like GOTO10
- M123- Back Spindle torque limit ON. Cuts the power of S2 by 25%
- M124- Back Spindle torque limit OFF. Sets S2 back to 100%
- M130- Z Sync on. Z2-Z1 synchronization on. Not available in MDI! M130 is automatically used in G650, you probably won't need it.
- M131- Z Sync off. Z2-Z1 synchronization off.
- M151- Gang tool post retract out of the way.
- M190- C2-C1 superimpose ON. Used to hold parts with both spindles and perform C1-axis work, C2 follows C1.

```

M48C0
M18C0
M190
G0C90. (MOVE C1 AND C2 FOLLOWS)
G98G1H180.F1000.
G0C0
M191

```

- M191- C2-C1 superimpose OFF
- M204- Thread Chamfering On. To have an exit chamfer while threading with G92. Use parameter #5130. The parameter is in one pitch intervals. So 1.0 would equal the exit to start one pitch from the threading end. #5130 can be .1 to 12.7. Chamfering can also be turned on or off by OPR switch #6.
- M205- Thread Chamfering Off
- M241- (macro) Move Z2 to safe position point. This position = 65mm. -If a long workpiece device is used then Z2 goes to Z0
-Command M241 in \$1 or after G600 in \$2
- M320- (macro)
Parts catcher used when not picking up with the sub spindle. A basket is mounted to the face of the sub spindle and Z2 will move up to catch the part, then back to dump it into the regular parts catcher. After M320 use M34 to dump the part. See M780 to setup the catcher in MDI. Only use for T4-T7 or special set operator setting 'Recovery Alarm' to use other tools with M320.

Also the basket to attach to the back spindle is an option.

M320X.2

X= Position to feed to before the basket is under the part. Used to eliminate chips in the catcher.

Z= The position of the basket of absolute Z2. Usually not used but can be used to fine adjust catching for problem parts. If not used 165mm or 6.496 is used.

F= Feed in IPM for movement of Z2

\$1

G0 X.5

(all chamfer commands if needed must be done)

(starting X must be more than M32X_)

M320X.15 or M320X.2Z6.4

G1 X-.05 F.002 (when X reaches X.15 then the catcher comes in)

M34(dump into ejector arm)

M700- macro) Like M7 but open the collet at any RPM. M700U.5 (U argument can be added to dwell after chuck opened)

M780- (macro) Use in MDI to set up the optional basket on the sub spindle cap when used as a parts catcher.

G-Codes

G0- Rapid move.

G00- If more than one axis is commanded each axis may move independently from the starting point to ending point of each axis. The path may not be a straight line.

G1- Linear interpolation

G01- Move in a straight line from one point to another

G2- Circular interpolation, clockwise. Right hand coordinates.

G02- You can also use I J K (distance to center point from the starting point of the arc) or R value.

G1X.4Z0F.002 or G1X.4Z0F.002

G2X.5W.05R.05 G2X.5Z.05K.05 (center of arc is Z+.05 from arc starting point)

G1Z__ G1Z__

G3- Circular interpolation counter-clockwise direction.

G03-

G4- Dwell

G04- G4 can be used 3 different ways...

G4 U1. (must have . or 1 = .0001)

G4 X1.

G4 P1000 (all three dwells = 1. second)

G9- Exact stop mode

G09- G9 will make the axis stop after the program line. Used when no radius is allowed on an outside or inside corner. G9 is a one shot g-code. See also M90.

G9 G1 Z.5

G17- Plane select- X-Y used when milling with "Y" on the Gang Tools

G18- Plane select- X-Z normally used for turning. G18 is when power on.

G19- Plane select- Y-Z

G25- Spindle Speed Change Detection off.

G26- Spindle Speed Change Detection on. The amount detection can be changed by programming and is saved for the next time used even after power off. Always G25 before a tool change or CSFM (G96). G96 can't be used if using G26. We use small tools so G26 is usually not used.

G26 P2000 Q10 R15

P2000 = 2 seconds before starting to check the speed.
(parm #4914)

Q10 = (+,-)10% of deviation or checking will not start
(parm #4911)

R15 = (+,-)15% of spindle speed before an alarm.
(parm #4912)

G28- Automatic reference return. Usually use the incremental command for each axis. "G28 U0 W0". You probably won't use G28 very often.

G32- Threading- G32 locks the feed-rate override at 100% and turns off the single block mode. Then G32 looks to find an exact spot on the spindle to start feeding. G32 is canceled by a G00 command. Its uses are for multiple angle threads (bone screws, worm gears) or we commonly use G32 when we tap a part with the front or back spindles. If you are cutting a special OD thread always start G32 on a Z move. If you start G32 on an X move, the spindle zero will pick up in a different Z position each time unless you program the X entry distance from the cut the same each time. Otherwise position "X" first then "G32 Z".

G32 can also adjust the spindle zero for double start threads.

G32 Q180000 = Q180. 180° from spindle zero.

tap example-

G600 (floating holder tap example 1/4-20 .5 deep)

T1300

M3S1=500

G0 Z-.05 T?? (position to center of part)

G32 Z.4 F.04 (locks feed-rate @100% and turns off single block.) Feed in at 80% of tap feed to pull head back)

Z-.05 F.05 M4 (feed-rate still locked @100% and no single block)

G4U.5

G0 Z-.1 (feed and single block are usable again)

M3

G34- Variable lead threading. This command is the same as G32 but has the "K" value for the amount of pitch change per revolution. See G32-

G34 X Z F.03 K.05

X= Position to go to in X

Z= Position to go to in Z

F= Feed rate in IPR

K.05= Amount of change in pitch per revolution. Can be growing pitch if plus value or reducing pitch if minus value. First rev will be F.03, 2nd rev is (.03+.05) 3rd rev is (.08+.05) and so on.

G40- Tool nose radius compensation CANCEL. Turns off G41 or G42

G41- Tool nose radius compensation LEFT. Usually used when front turning on Cincom machines in a Z positive direction. There are 8 quadrants of the radius to be used.

1= Boring

3= Back turning

4= Front turn (see manual for details)

8= Profile

0= Endmill

T500(Turning -Face and turn)

G0Z-.03T5 (set on offset screen R=tool radius and T=4(tip))

G42G1X.55Z0F.004 (Cutter Comp Right ON moving 2 axis)

X-.02

G41X.3,C.01F.002 (change to Cutter Comp Left)

Z.4

X.5,C.01

W.03

G40G0X.8T0 (cancel off the part)

G42- Tool nose radius compensation RIGHT. Not used as often turning. G42 could be used when traveling in a Z negative direction. See G41

G50- Coordinate system setting. Used to either set the position of an axis absolutely to a known value, or to move the position of an axis incrementally over a specified amount. The axis doesn't move but the position screen changes. Also used for spindle speed clamp.

G50 Z0 Makes current position of Z axis = 0, Z axis doesn't move.

Use "G50Z" in each program at the beginning to establish part zero.

G50 W-.5905 shifts the coordinate system over for temporary use then it would be shifted back by G50 W.5905

G50 SXXXXPYYYY Sets maximum RPM to XXXX and minimum RPM to YYYYY if in G96 mode. Works on the local spindle, \$1 or \$2.
You can Use G50S2000 or G50S1=2000

G43- (macro) Back Spindle control OFF

G44- (macro) Back Spindle control ON

This is automatically called in the G600/G640/G650 commands. If you programmed an operation from \$1 with T500K2 you may need G44.

G53- Machine Position moving. Move an axis to the systems Machine position from zero.

G53Z1.

G65- Calling a macro program. Same as M98 but we can pass information to the sub program using variables.

G66

G66.1 Special modal macro commands. See the Fanuc manual.

G67

G76- Canned threading cycle

G76 P010060 Q0020 R0005 (first G76 sets parameters for threading)

G76 X Z P Q F R (cuts the thread)

The first G76 isn't needed but is recommended.

- G76 P Q R

P010060 sets 3 things

- first 2 digits is the amount of finish passes - 01

- second 2 digits is % of the lead or pullout exiting the thread- 00. 00 = almost no angle at pullout and 99 = 9.9 leads away start out

- third 2 digits are the angle of infeed - 60. 0,29,30,55,60,80 are usable (0-90 is ok). The actual cutting infeed = (value/2)-.5 so a value of 60 = 29.5 infeed

Q0020 sets the minimum cut amount during threading .002 but no decimal point. (Q00200 for sub inch)

R0005 sets the cut amount of the last pass .0005 but no decimal point. (R00050 for sub inch)

The second G76 cuts the thread.

-G76 X.1876 Z.3 P0302 Q0010 F.05 (R-.002) FOR 1/4-20

X.1876 =Minor Dia. of thread

Z.3 or (W) = The ending Z of the thread

P0302 =Height of thread in radius (Maj-Min)/2 (.0302) (P03020 for sub inch)

Q0100 =Amount (.01) of the first cut. All the rest of the cuts are calculated. (Q01000 for sub inch)

F.05 =Feed-rate 20 TPI 1/20=.05

R = R is optional for tapered threading. R is the amount of difference in X from start to finish in Z. When cutting threads moving Z and X in a positive direction R is a negative value.

G80- Cancel G8? series codes. If only using G83 we don't need G80

G83- (option) Peck drilling cycle.

Example-
G0 Z-.05 T22
G83 Z1. F.001 R.03 Q04000 P0

Z1. =Z position of the bottom of the hole

F.001 =Feed in IPR or IPM

R.03 =Rapid from current Z position the R amount incrementally.

If starting at Z-.05 and R=.03 then the Z axis rapidly positions to "Z-.02" and after every peck Z retracts to the same position "Z-.02".
If you use "R-.03" it is the same as "R.03".

Q04000 =Peck amount -same as .04, decimal point is not allowed.
(Q04000 for sub inch / Q0400 for non sub inch)

P0 = Dwell at the bottom of the hole. You can just leave P off the command line if you want.

G84- (option) (macro) rigid tapping. Main Z1-S1 or sub Z2-S2. This is a Citizen macro version-not a Fanuc code.

M5	M25
T1300	T3300
G25	G25
G0X0Z-.05T13	G0Z-.05T33
G84 Z.5 R.03 D1 F.05 S500	G84 Z.5 R.03 D2 F.05 S500
G80	G80

or

(peck tapping)

M5
T1300
G25
G0X0Z-.05T13
G84 Z.3 R.03 D1 F.05 S500 H1
G84 Z.4 R.03 D1 F.05 S500 H1
G84 Z.5 R.03 D1 F.05 S500
G80

G84 Z.5 F.03125 D1 R.03 S500 (Q2.)

Z.5 =Z position of the bottom of the hole

D1 =spindle S1 or -1 for left hand taps (D2 for sub spindle)

F.03125 =Feed in IPR

R.03 =Rapid from current Z position the R amount incrementally. If starting at Z-.05 and R=.03 then the Z axis rapidly positions to "Z-.02". If you use "R-.03" it is the same as "R.03". Leave R out to start from current Z position.

S500=RPM

T1. = Dwell at the bottom of the hole. Use decimal point!

H1. = Use H1. if you are peck tapping. H1 does not use G80 in the macro before exiting the G84 macro.

Q2. =Speed to exit the tap. 2. = 2x500 or 1000 (this is an option)

G87- Cross Peck drilling cycle.

Example-
G0 X.6Z1.1 T8
G87 X-.55 F5. R.03 Q04000 (P0) H90. K2.
X-.55 =X position of the bottom of the hole
F5. =Feed in IPM

R.03 =Rapid from current X position the R amount incrementally. If starting at X.6 and R=.03 then the X axis rapidly moves to "X.54" and after every peck X retracts to the same position "X.54". If you use "R-.03" it is the same as "R.03".

Q04000 =Peck amount -same as .04 but decimal point is not allowed. Q04000 for sub inch / Q0400 for normal inch)

P0 = Dwell amount at the bottom of the hole. You can just leave P off the command line if you want.

H90. =Degrees to index for multiple features

K2. =Number of features or holes

G88- Cross rigid tapping. By default the tap uses phasing to keep rotational location of the tap. You can tap, do other work and come back and re-tap.

```
M82
T900
G50W-.59
GOX.8Y0Z1.5T9
G88 X0 R.05 F.05 D3 S500
G80
G50W.59
```

or (for peck tapping)

```
M82
T900
G50W-.59
GOX.8Y0Z1.5T9
G88 X.5 R.05 F.05 D3 S500 H1
G88 X.2 R.05 F.05 D3 S500 H1
G88 X0 R.05 F.05 D3 S500
G80
G50W.59
```

G88 X0 R.05 F.05 D3 S500 H1(Q2.)

X0 =Ending X position of tap
R.05 =Distance from position point to tap start point in radius. Leave R out to start from current X position.
F.05 =Feed rate.
D3 =spindle 3 (use D-3 for left hand taps).
S500 =RPM
T1. =Dwell at the bottom of the hole. Use decimal point!
H1. =Use H1 if you are peck tapping. H1 does not use G80 in the macro before exiting the G84 macro.
Q2. =Speed to exit the tap. 2. = 2x500 or 1000, (this is an option)

G90- Rough turning cycle used like G92

G92- Threading cycle. Straight infeed only. See also M204-M205

```
G92 X.22 Z.3 F.05 (R-.002) FOR 1/4-20
X.21
X.2
X.19
X.188
X.1876
X.1876
GO X.5 Z-.05 (ENDS G92 CYCLE. IF NO GO THEN ALARM!)
```

X.1876 = Each cutting pass

Z.3 or (W) =The ending Z of the thread

F.05 =Feed-rate 20 TPI 1/20=.05

R = R is optional for tapered threading. R is the amount of difference in X from start to finish in Z. When cutting threads moving Z and X in a positive direction R is a negative value.

G94- Facing cycle used like G90/92 but feeds down on X.

- G96- (option) Constant surface speed -on. You should turn off before any T???? code or you force the spindle up and down. So index to the tool then turn on. When finished turn off, and then index to the next tool. If you use G96 without S?= then the last G96 S? value is used.
- G97- (option) CSFM off. Maybe have at the beginning of your program as a safety command.
- G98- Feed commanded in Inch Per Minute designation.
- G99- Feed commanded in Inch Per Revolution designation.
- G113- Spindle Sync off. See G813
- G114.1- Spindle Sync On. See G814. These codes were added to be like Mitsubishi. You can use G814 or 114.1, your choice.
- G600's- The G600's codes are Citizen's way of making a complex multi axis machine easy to program. These codes all queue \$1 with \$2 so they are needed in both paths. There are arguments possible for each command that can change the axis movements as desired. See each code and its arguments below...
- G600- Cancel all G6xx commands. Also G600 is the default mode before the program starts and after the G999 command at the end of the program. G600 will queue with both lines \$1-\$2. G600 automatically puts \$1 and \$2 into G99 mode. G600 in effect in MDI.

```

$1          $2
G600       G600
           T3100
           GOZ-.05T31
           G1Z.1F.002
           GOZ-.1T0
!1!2L1     !1!2L1

```

or

```

$1          $2
G600       G600
T3100K2
GOZ-.05T31
G1Z.1F.002
GOZ-.1T0
!1!2L1     !1!2L1

```

G600 command options...

```

$1      G600    (no arguments are allowed in $1)

$2      G600    (No arguments means Z2 moves away to Z2 home position)
        G600W0 (W0 means leave Z2 at the current position)

```

- G660- Machining mode change to use the front ID tools (T11-T14) and also the back side (T51-T54) at the same time. G660 will queue with both lines \$1-\$2 then set auto queuing between tool calls of T11-T51, T12-T52, and T13-T53. After G660 \$1 controls X1-Y1-Z1-C1 and \$2 has Z2-C2. The Z2 position is set to program the sub spindle part zero to the back side drill zero.

G660 command options...

```

$1 G660 (no arguments are allowed in $1)

$2 G660 (no W0 means Z2 moves away to Z2 home position)
G660W0 (W0 means leave Z2 at the current position)

```

Example...

```

$1          $2

```

G660	G660
T1100	T5100
G0 Z-.05 T11 (peck drilling 8 sec)	G0Z-.05T21 (peck drill 15 sec)
T1200A2. (queues with T5200)	T5200
T1300 (queues with T5300)	T5300
G600	G600

G650- Machining mode change to superimpose Z2 on to Z1. G650 will queue with both lines \$1-\$2 then make Z2 follow Z1.
Example -when we pick up a part and chamfer we while we have the back spindle collet closed on the part.

Note- Any shift, "G50W-.125", on Z1 must be canceled, "G50W.125", before G650 or G650 won't find the face of the part!

G650 command options...

\$1 G650 (no arguments Z2 is not superimposed)

\$2 G650 (no W0 means Z2 moves away to Z2 waiting position then moves)

G650M1 (M1 superimposes Z1 coordinates onto Z2)

G650W0 (W0 arguments Z2 does not moves from current position)

G650Z-.1 (Z_ arguments Z2 moves to Z_ Z1 position)

Example...

\$1	\$2
G600	G600
T100(cut-off)	(eject part)
G50W-.125	
!2L100	!1L100
M3S1=3000	
M24S2=3000	
G0X.5Z2.T1	
G50W.125	
G650	G650
	G0 Z-.1(bring Z2 to Z1-.1)
!2L650	G1Z1.F.05(feed over)
G50W-.125	M15(close collet)
(moving Z1)	!1L650(let \$1 continue)
"	
G1X-.1F.001	
G600	G600(bring Z2 back)
G50W.125	M25
S1=1500	
M7	
G0X-.1Z-.005T0	
M56	
G999	G999
N999	N999
M2	M2
M99	M99

G813- (macro) Same as G113, to turn off spindle sync.

G814- (macro) Spindle sync on. See also G113 and M77

M3S1=300
M24S2=300
G814 R0 (use R0 to force phasing S1-S2)
M77
M3S1=4000

Example to pick-off a round part with CSFM...

(\$1)	(\$2)
T100	
G50W-.125S5000 (PO 5000 RPM)	
GOX1.1Z4.T1	
M3S1=300	
M24S2=300	
G814	
G96 S1=200	
G650	G650
!1!2L2	G98G1Z1.F50.
G1X-.05F.0015	M77
X-.1F.003	M15
G50W.125	!1!2L2
G97	
G813	
M5	
M25	
G600	G600

G899- GB Phase adjustment. Used for hex stock phase alignment of front and back spindles. You manually put the stock through the front spindle collet, guide bushing and back spindle collet then from MDI command G899. This will “teach” the machine what rotational position for the back spindle will corresponds to C0 on the front spindle. You must use a collet with a keyway so the collet will not rotate in the back spindle during use.

G899 (K2) (use K2 if you do not want the main spl chuck to close)

G999- (macro)

Commands between G999 and N999 on each program path will execute at the end of the part program when:

In SINGLE CYCLE MODE.

In CONTINUOUS mode but the work counter is full.

In CONTINUOUS mode and other alarms come up. (no bar remnant, no bar, work counter full, ect.).

Often \$1 doesn’t need to do anything to complete the backwork on the part so no codes are required between G999 and N999. Sometimes \$1 tools are required to complete backwork on a part, it is still possible to machine one part totally complete. Whatever programming is required for \$1’s contribution to the backwork machining of a part is added after G999 with appropriate queueing codes. It is very important to also code in machine moves necessary to get the front spindle tools back to the same position they would be after partoff in normal operation.

It is a good practice to have programming code for a given feature only appear once in a program. If programming for a feature appears twice in the program; once in the part program and another time between G999 and N999 for example, any edits to the program to improve the feature would need to be performed on both sets of code. It is easy to forget to edit both sections of code. If this happens you end up with most of the parts being machined one way and occasionally a part is machined differently.

Example:

\$!	\$2
	N10
FRONT SPINDLE CODE FOR PART	#100=15
	GOTO2000 (CODE FOR BACKWORK STARTS ON LINE 2000)
	N15
G650	G650
(PART CUT OFF)	(PART PICKOFF)
G600	G600
M5	M25
M8	

/M98P5000(BAR LOAD SUB PROGRAM)

M9

G999

G999

#100=99

GOTO2000

N99

N999

N999

M2

M2

M99

M99

N2000(BACKWORK CODE)

MACHINE BACK FACE OF PART HERE

M34(DUMP PART)

GOTO#100

M99

The code between G999 and N999 only executes in 1 cycle mode and if the finish part switch (OFFSET button, OPR softkey, Last Part is 1) is active. During continuous operation the \$2 program executes down to line N10, reads the #100=15 line and enters "15" into variable #100 then jumps to line N2000 which is the start of the backwork code. After the part's back face is finished and dumped into the finished part chute, the GOTO#100 line reads as "GOTO15" since variable #100 = 15 at this stage of the program execution. If the machine is put into 1 cycle mode and the "LAST PART" switch is "1", then the code between lines G999 and N999 execute for both \$1 and \$2. \$1 doesn't have any code so it does nothing while \$2 executes the lines "#100=99" which changes the number stored in variable #100 to "99" and then \$2 jumps to line N2000. The back face of the part is machined and the part is dumped into the finished part chute then the "GOTO#100" line is read at "GOTO99" (variable #100 contains "99") which jumps the program to line 99. The machine then gets to M2 (end of main program) for both \$1 & \$2, then executes M99 (rewind). The controller goes to the start of the program and the green cycle start button turns off.

!1!2L1-Queuing between lines \$1-\$2. Used as waiting commands to make \$1 wait for \$2 or \$2 wait for \$1.

!1= line \$1

!2= line \$2

L1= queuing reference #. 1-9999. Also "L" does not have to be in order.

Example...

O1234(queuing example)

\$1 \$2
(10 seconds of CNC commands)

!1!2L5 !1!2L5 (\$2 and \$1 wait until on !1!2L5 then continue)

!1!2L1 !1!2L1 (\$2 and \$1 wait until on !1!2L1 then continue)

O1234(queuing example does the same as above)

\$1 \$2
(10 seconds of CNC commands)

!2L5 !1L5 (\$2 and \$1 wait until on !L5 then continue)

!2L1 !1L1 (\$2 and \$1 wait until on !L1 then continue)

O1234(queuing example does the same as above)

\$1 \$2
(10 seconds of CNC commands)

!L5 !L5 (\$2 and \$1 wait until on !L5 then continue)

!L1 !L1 (\$2 and \$1 wait until on !L1 then continue)

T-Tool selection. Each tool call will rapid to position point of the old tool and then to the position point of the new tool. A four digit code is a tool call and a two digit code is an offset. If you have arguments on the T-call line they have different meanings depending on

which set of tools are called. The A2-20 also does a calculation when using the gang. It will take the largest diam. setting from the tool set page and add this to each tool while indexing, then come down to the position point for the tool.

T100	tool call	(T0100)
T101	tool call with offset	(T0101)
T1	offset call	(T01)
T0	cancel offset	(T00)

T0100-T900 (Front side Gang tools)

T0200 X Y Z Q1 H K2 E

T700 K2

X.3 = Use X to position X after the tool call.

Y.3 = Use Y to position Y after the tool call.

Z.3 = Use Z to position Z after the tool call.

Q1 = Quick index. This will not move to safe position then index, It will move from where it is to the new tool in a straight line. Be careful.

H.5 = Temporary change position point for just this one index.

K2 = Used to call the Z2 axis as being programmed. If 'T3100K2' in \$1 then X1-Y1-Z2-C2 are programmed in \$1. K2 is canceled on any other Tool call. Also if using 'T100K2' to 'T600K2' the Z2 zero position is automatically set to the back side of the tool knowing the sub spindle with part face as zero. If 'T700K2' to 'T900K2' is used then the Z2 zero position is automatically set to the center of the live tool knowing the back spindle with part face as zero.

\$1 T300 \$1=X1Y1Z1C1 \$2=X2Z2C2

\$1 T300 K2 \$1=X1Y1Z2C2 \$2=X2Z1C1

\$2 T300 alarm -not allowed

\$2 T300 K2 \$1=X2Z1C1 \$2=X1Y1Z2C2

E90.= Use E to save cycle time. The C axis will go to E90. while changing tools. Do not use M5 and do not use M18

T1100-T1400 (front side ID tool stations 1.0" bore)

Be careful as T11-T14 assumes the stock is clear and does not move to a safe position up to index above the OD turning tools. T11-T14 takes the quickest X Y path to center the tool.

T1100 X Y Z A E

X Y Z = Move to axis position with the tool call, saving time.

E90.= Use E to save cycle time. The C axis will go to E90. while changing tools. Do not use M5 and do not use M18

A= If in mode G660 then argument 'A' can be used. Tool position is called by splitting the difference of for example T1100-T5100. T1100 has positioning parameters and T5100 has positioning parameters that may be a little different. If in G660 and T1100 then split the error of both T1100 and T5100 positioning. A1.= Then use T1x00 positioning settings for tool centering. A5.= Then use T5x00 positioning settings for tool centering. (If used you need to use the '.'. So use 'A1.' 'A5.'

T3000 (back spindle X2 center to main spindle)

T3000 Q1 Z W

Q1 = No back spindle movement. If no Q1 then Z2 moves back to a safe position then moves to spindle center.

W.5 = This will move the Z2 incrementally back then move to center. If no W is used then Z2 goes back to a safe position then indexes to the spindle center.

Z.5 = This will move the Z2 to Z position after centering the spindle. Use Z.5Q1 and Z2 will not go back first.

T3100-T3400 (back ID tool block 1.0" bore)

T3100 Z W X Q1 Q3

W.5 = This will move the Z2 incrementally back then move to center. If no W is used then Z2 moves back 5mm or .1968.
 Z-.2 = This will move the Z2 to Z position after centering the spindle. Use Z-.2Q1 and Z2 will not go back first.
 X.25 = This will move the X2 to X position after centering the spindle.
 Q1 = Quick index. This will not move Z2 to safe position then index, It will move from where it is to the new tool in a straight line. If Q1 is not specified the X1Y1 axis will start to move when Z2 moves back 5mm or .1968.
 Q3 = Force Z2 to move to back position point then index the tool.
 No Q = Z2 moves back incrementally 5mm .1968 then changes the tool.

T5100-T5400 (back side of main Gang ID tool block 1.0" bore shared with T11-T14)

T5100 Z W X Y Q1 Q3 E

W.5 = This will move the Z2 incrementally back then move to center. If no W is used then Z2 moves back 5mm or .1968.
 X Y Z = Move to axis position with the tool call, saving time.
 Q1 = Quick index. Same explanation as T3100- T3400 above.
 Q3 = Same explanation as T3100- T3400 above.
 no Q = Same explanation as T3100- T3400 above.
 E90.= Use E to save cycle time. The C2 axis will go to E90. While changing tools. Do not use M25 and do not use M48

Note- Any other codes must be before the T command on the line. Also if you're using G96 you must cancel with G97 before any T???? command.

M3S1=2000

T202

G96 S1=300

"

"

"

G97G40

T303

G96 S1=250

"

"

G97G40 or G97S1=1000

(Option - I/K/R won't work with direct drawing). If parameter #3453 bit 0 = 0 then Auto Corner (I/K/R) will work and direct drawing will not work.

X K- used to do trig for us

X I- used to blend radii on angles

X R- used to blend chamfers on angles

X?? K.03- (option) Automatic corner chamfer. "K" is for X-Z Usually used when front turning.

G0 Z0

G0 Z0

G1 X-.04

G1 X-.04

X.19

X.25 K.03

X.25 W.03

Z1.(has to move more than .03 or alarm)

Z1.

Z?? I-.02- (option) Automatic corner chamfer. "I" is for Z-X. Usually used when back turning.

G1X.25

G1X.25

Z.98

Z1. I-.02

X.21 Z1.

X.125(has to move more than .02x2 or alarm)

X.125

Z1.

Z1.

X?? R.02- (option) Automatic corner rounding. You can use this function or Direct Drawing Input, but not both. Change by parameter.

G0 Z0	G0 Z0
G1 X-.04	G1 X-.04
X.21	X.25 R.02
G2 X.25 W.02 R.02	Z1.(has to move more than .02 or alarm)
G1 Z1.	

(Option -Direct drawing input/ I/K won't work with direct drawing). If parameter #3453 bit 0 = 1 then direct drawing will work and Auto Corner (I/K/R) will not work.

X ,A- used to do trig for us

X ,R- used to blend radii on angles

X ,C- used to blend chamfers on angles

X ,A,R-

G18(,R,A works with the current plane)

G1X0Z0

X.05,R.01 (blend a radius at the beginning of the angle)

X.25,A30.,R.01 (if Z is not known then ,A will figure Z)

Z.5