USER SCREENS

OPERATION & USE

FAGOR 8055iT

JULY 2017
REVISION 2.15

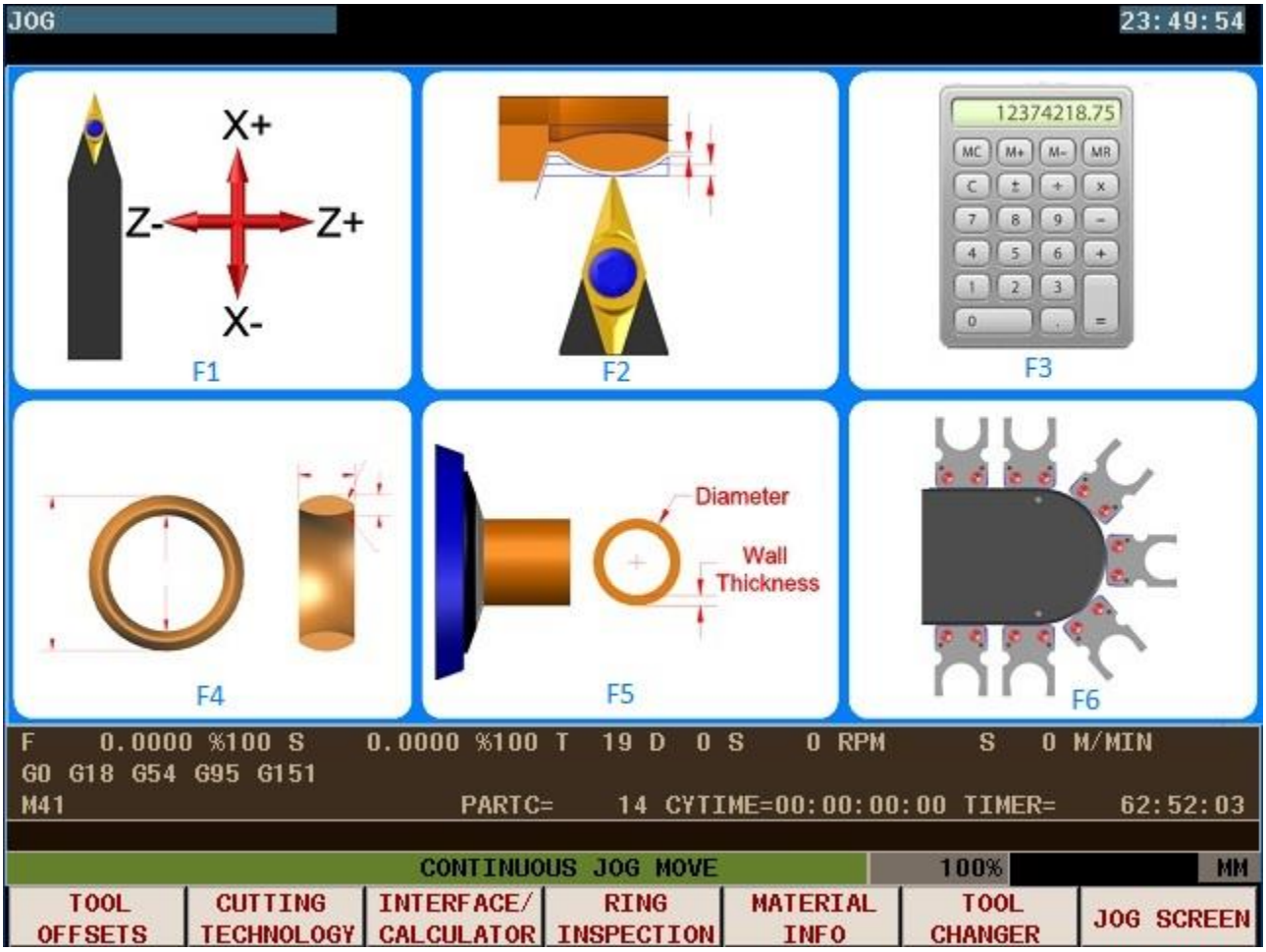
Table of Contents

Initial User channel Screen		3
F1 Tool Offsets		4
F1	X-Z Tool Offsets	5
F2	Facet Tool Radius	6
F3	Carousel Tool Shift	7
F4	Turret Tool List	8
F5	Pre-Setting Live Tools	9
F6	Back	9
F7	Jog Screen	9
F2 Cutting Technology		10
F1	Turning Feed Data & explanation of the material library	11-12
F2	Turning Tool Data	13
F3	Milling Feed Data & Live Tool Drilling Data	14-15
F4	Milling Tool Data	16
F5	Turning RPM	17
F6	Milling RPM	18
F7	Back	
F3 Interface & Calculator		19
F1	Dome Radius	20
F2	Round Stone Sizes	21
F3	No Selection	
F4	No Selection	
F5	No Selection	
F6	No Selection	
F7	Back	
F4 Ring Inspection		22
F1	Calculate Ring Size	
F2	Calculate Groove Depth	
F3	Adjust Thickness	
F4	Tool Offsets	
F5	No Selection	
F6	Back	
F7	Jog Screen	
F5 Material Information		23
F1	Tube	
F2	Solid Bar	
F3	Blanks	
F4	Graphics Refresh	
F5	Weight Information	
F6	Back	
F7	Jog Screen	
F6 Tool Changer		24
F1	Open Door	
F2	Close Door	
F3	Advance Tools	
F4	Retract Tools	
F5	Home Carousel	
F6	Last Position	
F7	Jog Screen	

USER CHANNEL

The User channel provides a group of screens that provide data on tooling, speeds and feeds, ring sizes and other useful data for controlling the machining operations.

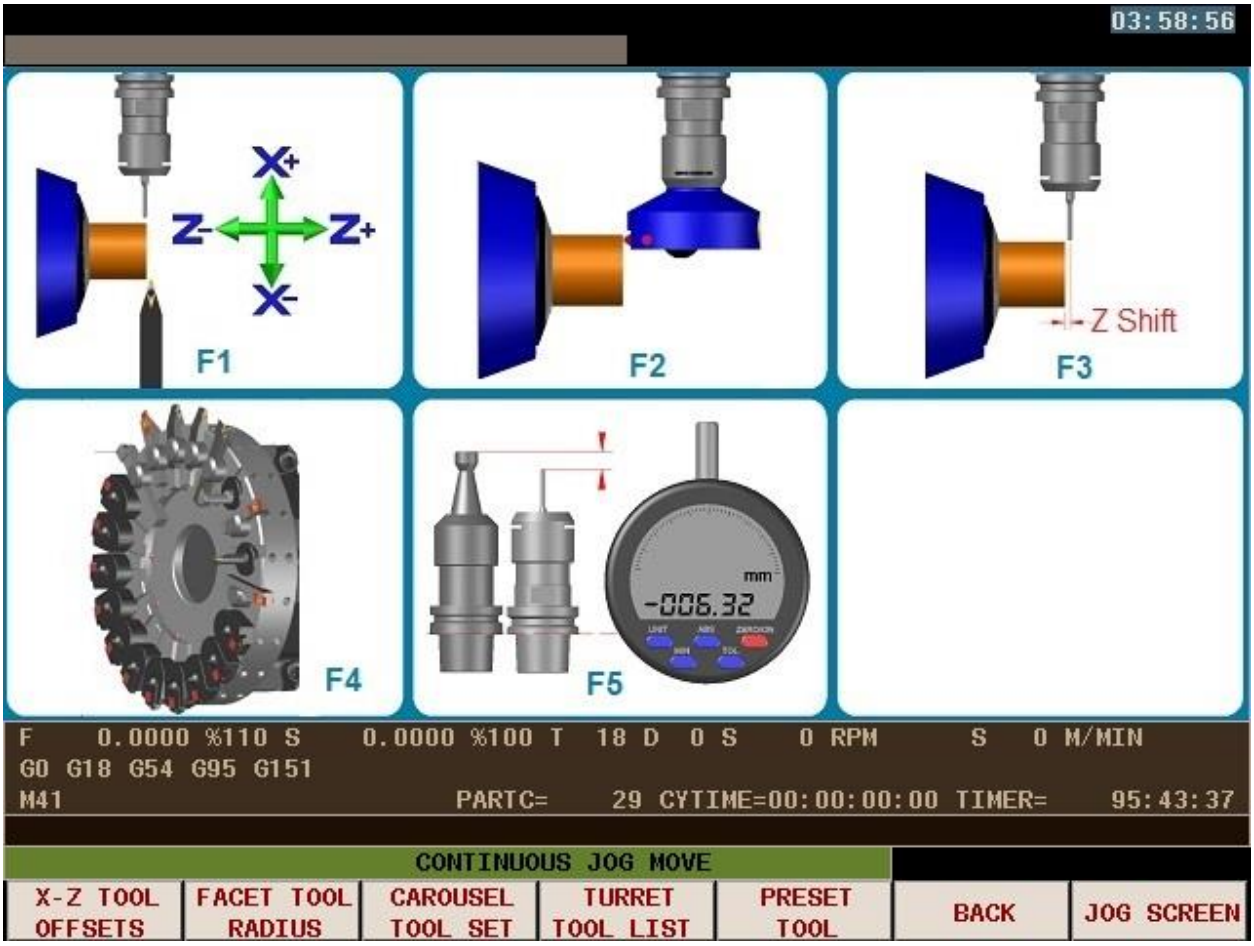
The USER key brings up this screen



The 7 function keys along the bottom provide the method of selecting each screen choice.

Each selection is described in the following sections.

F1 TOOL OFFSETS



There are 5 selections available:

F1 **X-Z TOOL OFFSETS**. This allows the adjustment of offsets to fine tune the size of the finished rings.

F2 **FACET TOOL RADIUS**. This simplifies setting the tool radius values on faceting tools.

F3 **CAROUSEL TOOL SHIFT**. The machine uses 1 Z offset for the live tool, this allows the quick setting of all 18 offsets to the same value.

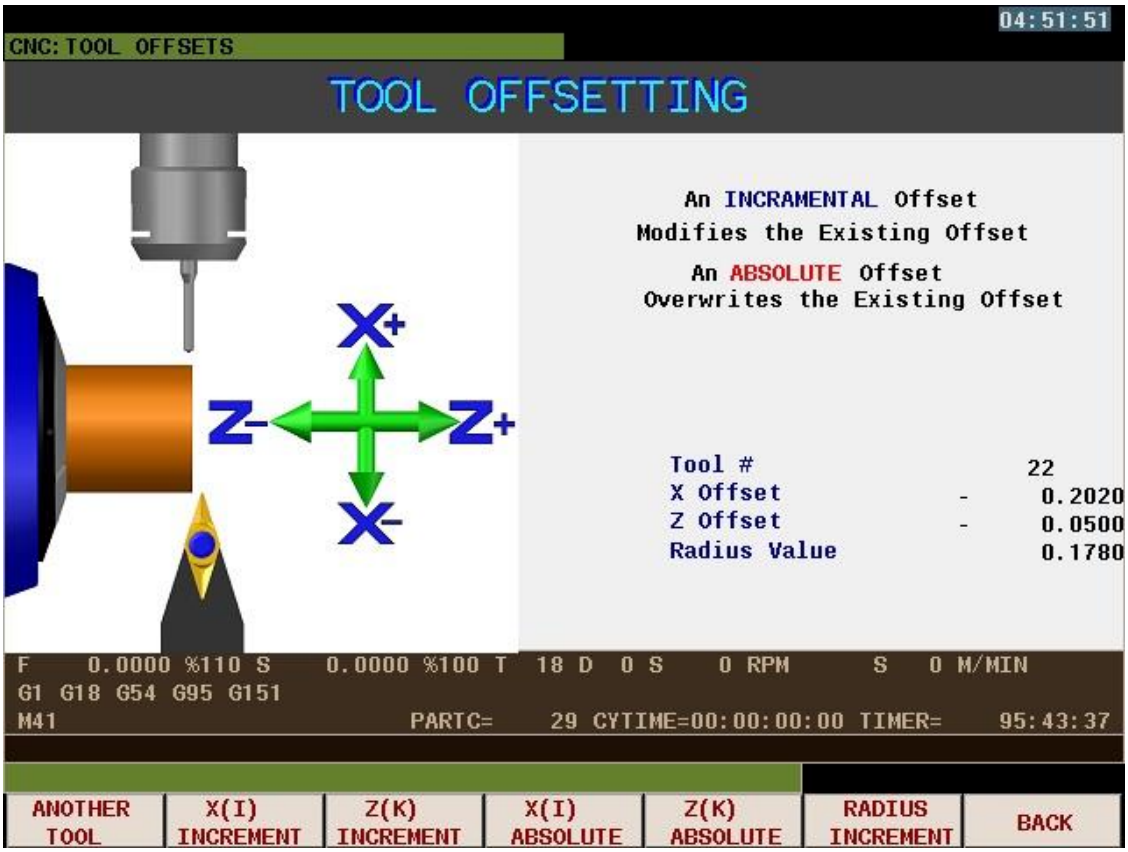
F4 **TURRET TOOL LIST**. This provides a list of tools and which position they occupy on the turret disk.

F5 **PRESET TOOL**. This screen provides the input for the live tool preset data. (Requires the tool presetter)

F6 **BACK**. This key returns to the previous screen.

F7 **JOG SCREEN**. This key returns to the jog screen.

F1 X-Z Tool Offsets (Tool offsets)



This screen allows the adjustment of X & Z offsets for each tool, both live tools and turning & boring tools.

F1 **ANOTHER TOOL** This allows the selection of a different tool after the initial tool has been selected

F2 **X(I) INCREMENT** This provides a method of offsetting an X axis value that is added or subtracted to the existing offset.

F3 **Z(K) INCREMENT** This provides a method of offsetting an Z axis value that is added or subtracted to the existing offset.

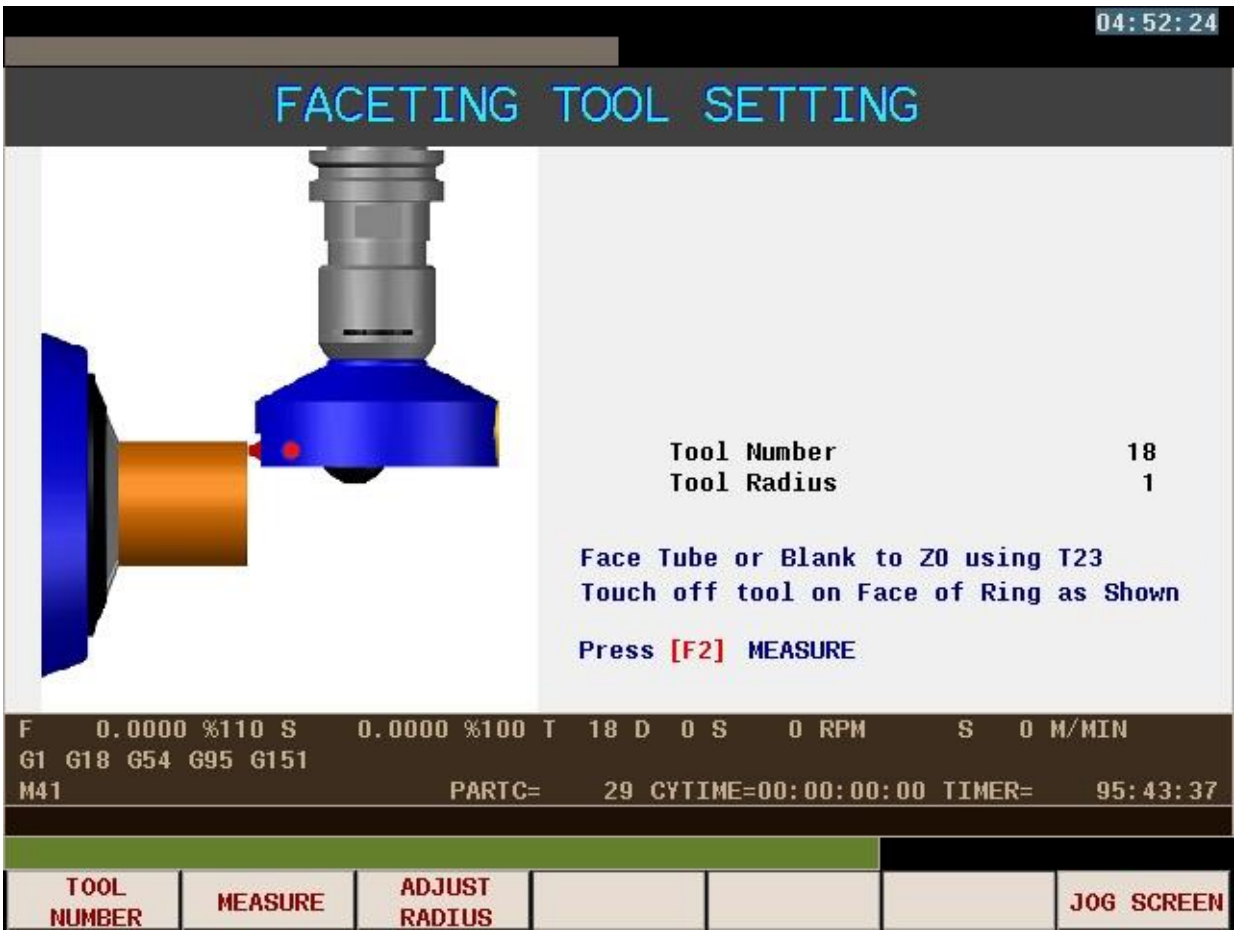
F4 **Z(K) ABSOLUTE** This provides a method of inputting a specific value into the X Axis offset for the selected tool.

F5 **Z(K) ABSOLUTE** This provides a method of inputting a specific value into the X Axis offset for the selected tool.

F6 **RADIUS INCREMENT** This allows a value to be added or subtracted to the selected tool radius value.

F7 **BACK**. This key returns to the previous screen.

F2 FACET TOOL RADIUS (TOOL OFFSETS)



This screen allows the setting of the radius value for a large faceting cutter that cannot be measured easily.

F1 **TOOL NUMBER** Use this key to set the tool number of the tool being set.

F2 **MEASURE** This calculates the radius value and enters it into the tool table.

F3 **ADJUST RADIUS** This allows a value to be added or subtracted to the selected tool radius value.

F7 **JOG SCREEN**. This key returns to the jog screen.

OPERATION

Face the tube or blank with tool T23 to Z0.

Then start the live spindle at 8000 rpm by pressing the start button on the left soft key

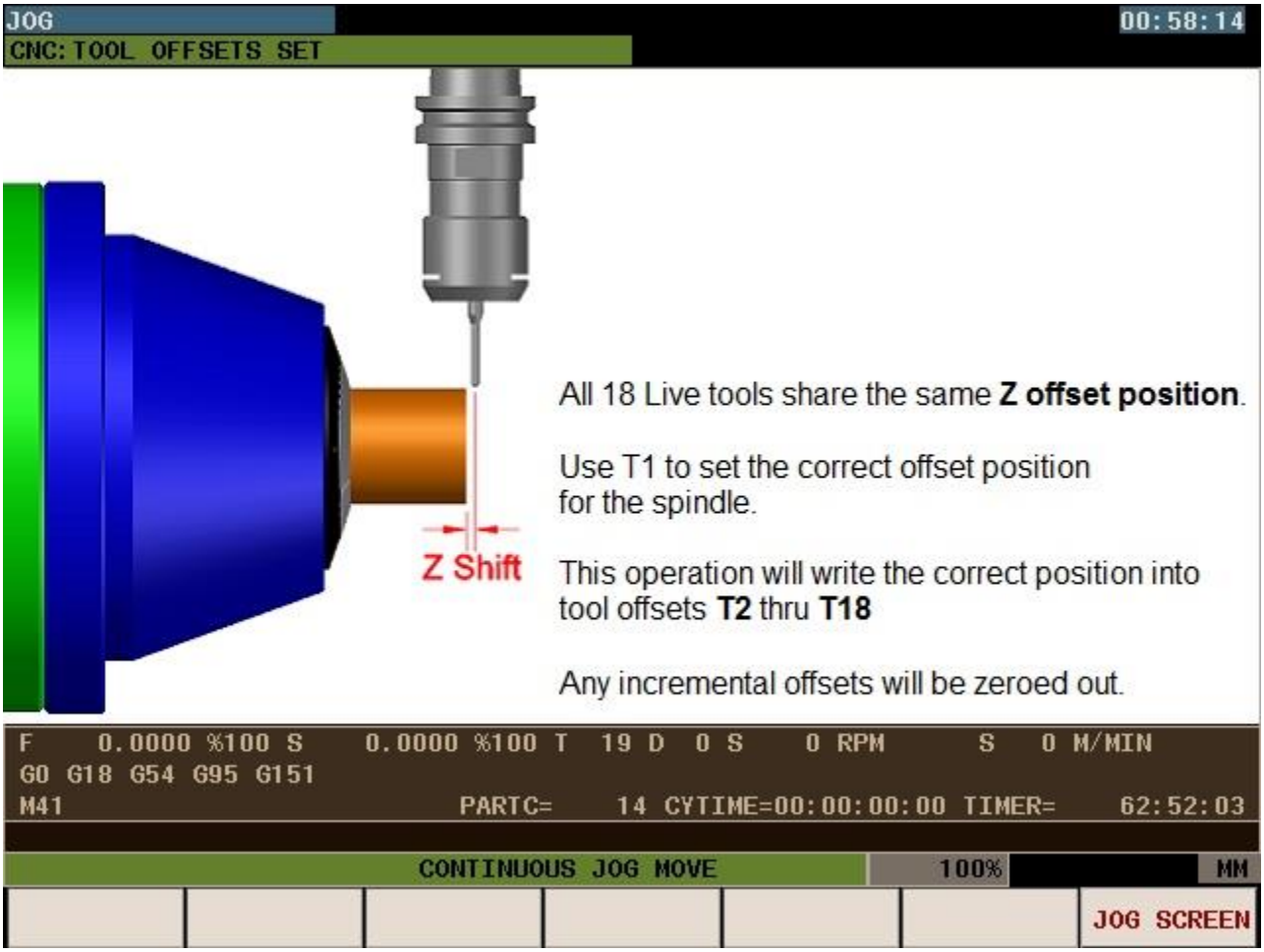
Touch off the tool on the previously machined face.

Enter the tool offset number to be used.

Press F2 MEASURE, this will load the correct value for the radius of the facet cutter.

This value is **critical** to the correct operation if the faceting cycles.

F3 CAROUSEL TOOL SET (Tool offsets)



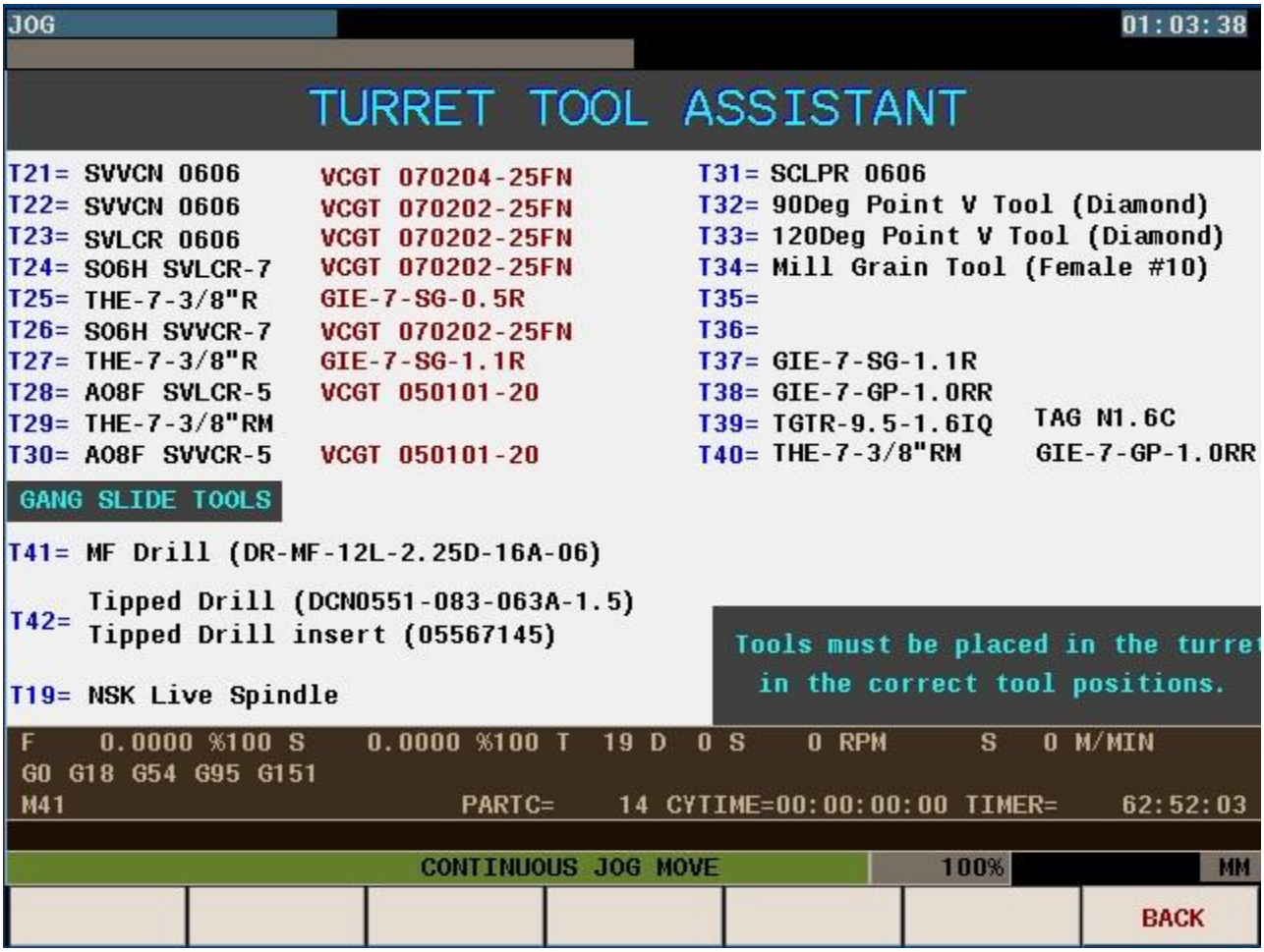
This key uses the value in the **T1 Z offset** to change all the values for tools **T2** through **T18** to the T1 Z offset value.

This operation also clears any K offset value in tools T1 through T18.

As the live spindle is fixed and the tools all are in the same spindle, all Z offset values are the same.

Only X offset values change depending on the length of each individual tool.

F4 TURRET TOOL LIST (Tool offsets)

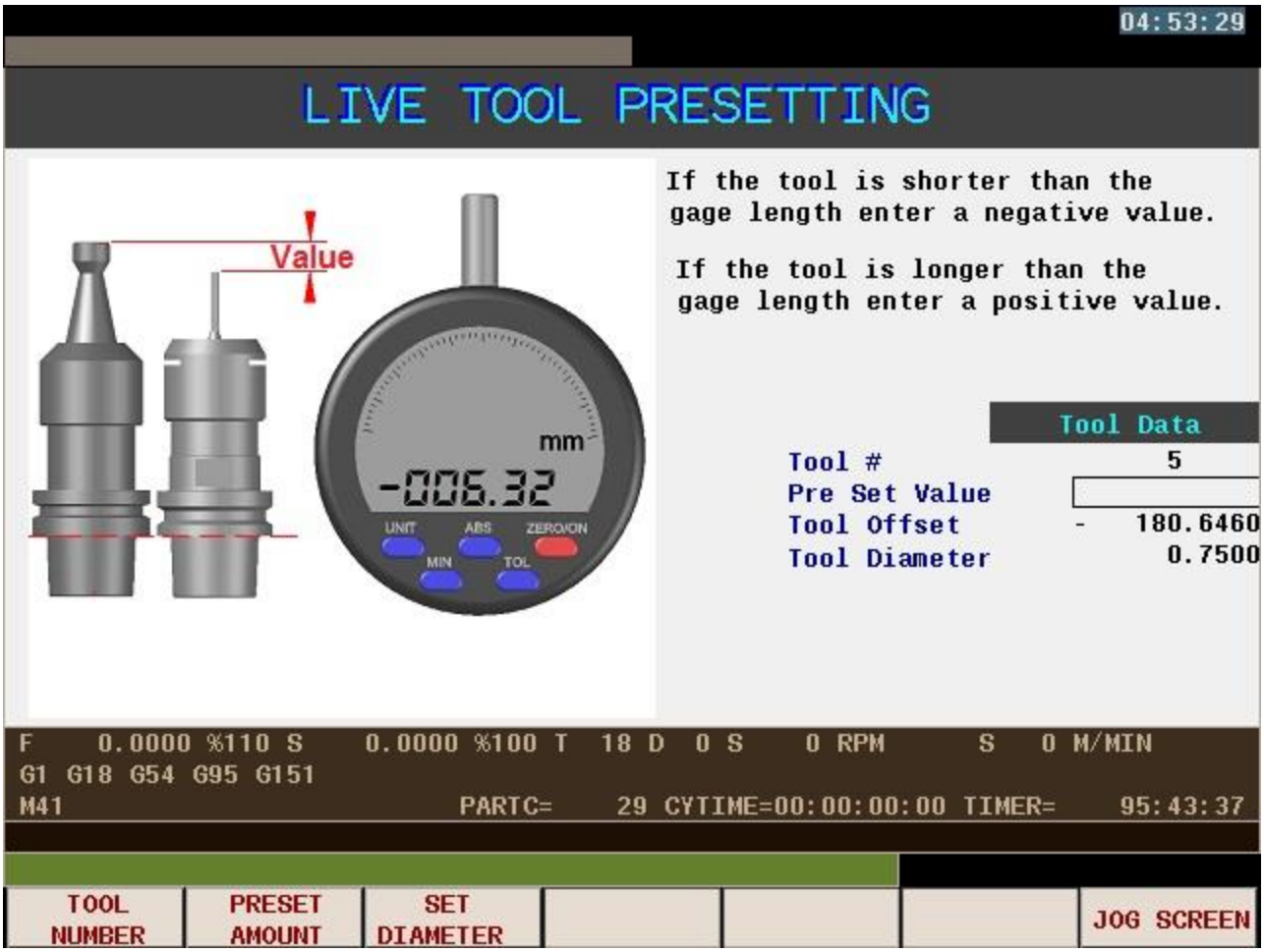


This screen displays the tool numbers and descriptions of each tool in the turret.

This is for informational purposes only.

This screen can be edited to reflect different tool requirements and placement.

F5 PRESET TOOL (Tool offsets)



This screen allows the setting of the tool length value of the live tools. This process allows a faster way to set very small tools without the chance of breaking tools when touching off on material in the machine.

This method of tool setting greatly decreases the time required to set tools.

F1 **TOOL NUMBER** Use this key to set the tool number of the tool being set.

F2 **PRESET AMOUNT** This is the value measured on the tool presetter.

F3 **SET DIAMETER** This allows a Diameter value to be added to the tool table for the tool selected.

F4 **NO SELECTION.**

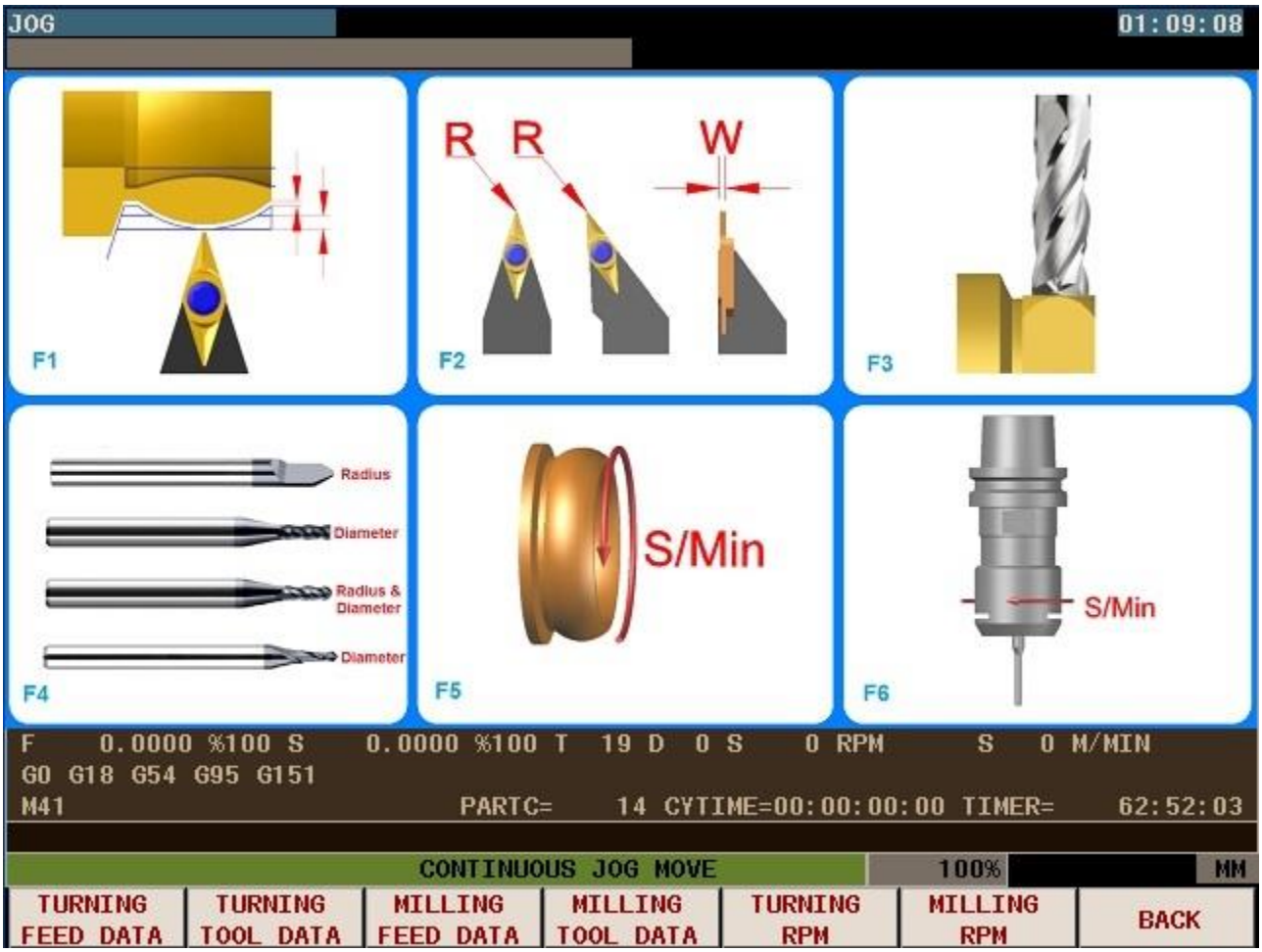
F5 **NO SELECTION.**

F6 **NO SELECTION.**

F7 **JOG SCREEN**. This key returns to the jog screen.

Main Screen F2 Cutting Technology

This group selection provides screens to set up cutting speeds and feed values for the different tool types.



There are 6 selections available:

F1 **TURNING FEED DATA.** Feed rates for the different turning tools

F2 **TURNING TOOL DATA**. Geometry data for the tool graphics and correct operation of cycles.

F3 **MILLING FEED DATA**. Feed rates for the live milling tools.

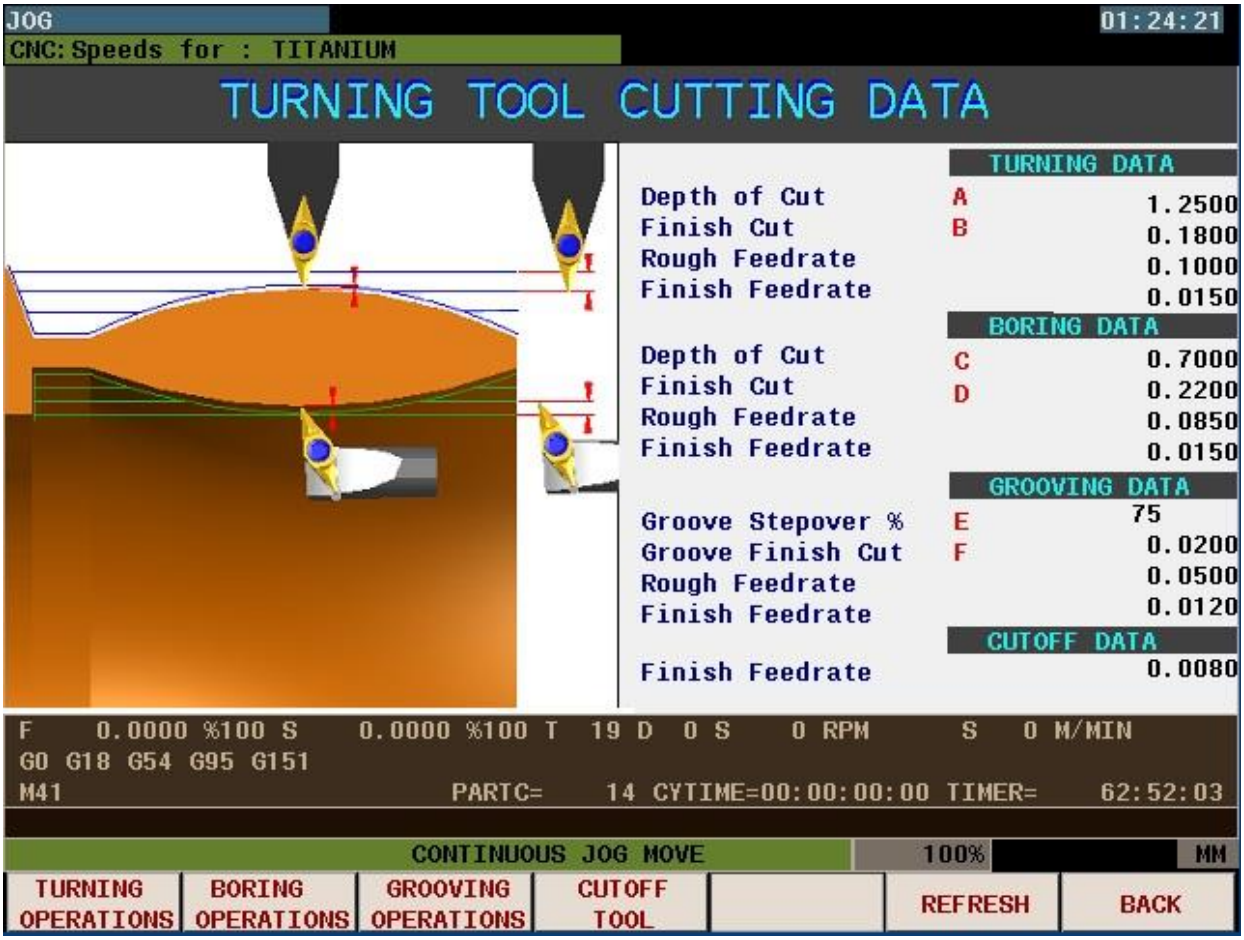
F4 **MILLING TOOL DATA**. Tool radius values and tool descriptions for the tool carousel.

F5 **TURNING RPM**. Main spindle speeds for different operations & materials.

F6 **MILLING RPM**. Live spindle speeds for different operations & materials.

F7 **BACK**. This key returns to the previous screen.

F1 TURNING OPERATIONS (Cutting Technology)



There are 6 selections available:

F1 **TURNING OPERATIONS.** This screen shown is displayed when turning operations is selected. The questions toggle through the values listed.

Depth of cut is the depth used for roughing cycles, to rough cut the profile
Finish cut is the amount left in a 2 pass cycle for the final finish cut.
Rough feed rate is the feed used for a roughing cycle of a 2 pass initial cut.
Finish feed rate is the feed used for the final pass in a roughing cycle or 2 pass cycle, or the federate used for a 1 pass cycle.

F2 **BORING OPERATIONS.** The same questions are asked when setting the boring feed rates.

F3 **GROOVING OPERATIONS**. The grooving tools need to know the amount of step over as a % of the tool width.
There is a finish cut amount for the sides of the groove
There is a rough & finish feed rate for the grooving cycles.

F4 **CUTOFF TOOL OPERATION**. Tool radius values and tool descriptions for the tool carousel.

F5 No selection

F6 **REFRESH.** This reloads the values in memory so that they can be viewed.

F7 **BACK**. This key returns to the previous screen.

F1 TURNING OPERATIONS (Cutting Technology)

Overview of Cutting Feeds & Speeds

The feeds and speeds can be adjusted on these user screen pages.

The File 90007.PIT or Material Library is the location within the software where the speeds and feeds are located for different materials.

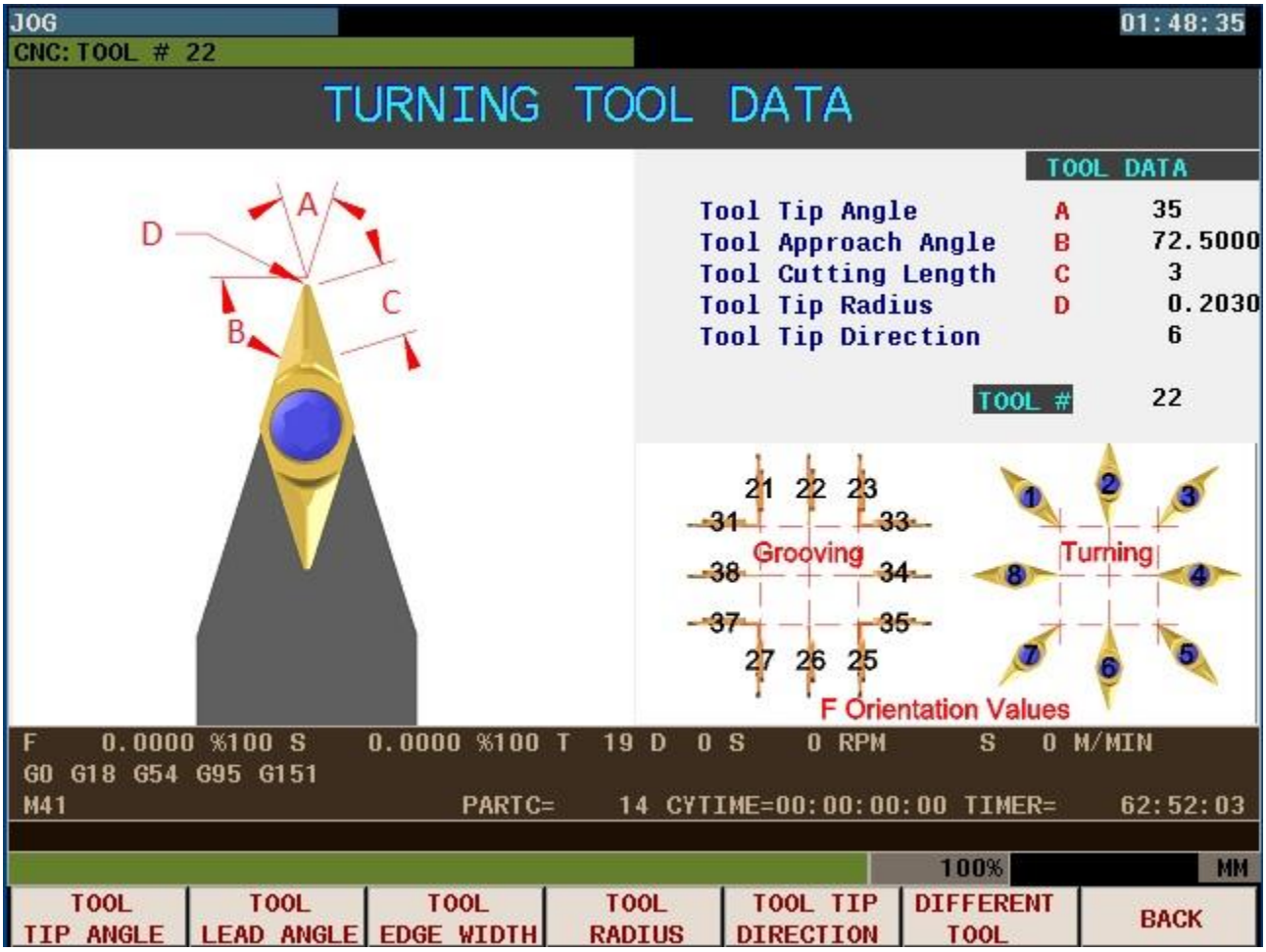
Adjusting the speeds & feed rates in the user screen does not load the speeds & feeds into the material library.

If the speeds and feeds are adjusted in the user screens to fine tune the cutting performance, the values should be transferred to the appropriate material file in the material library.

Shown below is a typical material library file.

```
(SUB 11001); ================= 10KT YELLOW GOLD
(P2036=701)
(MSG P2036)
;TURNING INFO
(P1163 = 0.015)  ; Finish Turning feedrate
(P1164 = 0.05)   ; Rough Turning feedrate
(P1169 = 0.18 )  ; Outside Finish amount
(P1170 = 0.65 )  ; Turning Depth of Cut
(P1215 = 0.007)  ; Form groove feedrate
(P1167 = 0.015)  ; Cutoff feedrate
;GROOVING INFO
(P1165 = 0.05)   ; Rough groove feedrate
(P1166 = 0.012)  ; Finish groove feedrate
(P1168 = 75 )    ; Groove cut %
(P1171 = 0.02 )  ; Wall allowance
(P192 = 100)     ; Form tool dwell time
;BORING INFO
(P1200 = 0.7)    ; Inside Depth of Cut
(P1179 = 0.22)   ; Inside finish amount
(P1220 = 0.085)  ; Rough Feedrate
(P1221 = 0.015)  ; Finish Feedrate
;MAIN SPINDLE INFO
(P1173 = 3400 )  ; Max RPM
(P1174 = 200)    ; Surface speed
(P1178 = 600)    ; Mill grain RPM
(P1187 = 165)    ; Cutoff tool RPM
(P1203 = 1200)   ; Form Tool RPM
(P1217 = 1200)   ; Threading RPM
;STATIC DRILLING INFO
(P1229 = 0)      ; 0= NO DRILL 1=MF DRILL 2=TIPPED DRILL
(P1222 = 800)    ; Drill RPM
(P1224 = 1.5)    ; Peck depth
(P1225 = 0.05)   ; Drill feedrate
;MILLING INFO
(P1159 = 0.005)  ; Drilling Feedrate
(P1160 = 0.05)   ; Clearance Distance drilling
(P1161 = 0.012)  ; Rough milling Feedrate
(P1162 = 0.008)  ; Finish milling Feedrate
(P1190 = 400)    ; Milling Surface speed
(P1192 = 0.002)  ; Feed to depth in X
(P1193 = 0.75)   ; % of Tool Diameter for step over
(P1197 = 0.25)   ; Mill depth of cut
(RET)
```

F2 TURNING TOOL DATA (Cutting Technology)



Each tool has its own graphic to help explain the values.

These values are used by the controller for tool graphics in simulate mode and also to be able to calculate tool radius compensation so that the correct profile is produced.

There are 7 selections available:

F1 **TOOL TIP ANGLE.** **A** This is the tip angle of the insert.

F2 **TOOL LEAD ANGLE.** **B** This is the angle the tool is set at, here is a same of math to see how we get 72.5^{O} The tool is a 35^{O} tip angle. 35/2=17.5 90-17.5=72.5

F3 **TOOL EDGE WIDTH**. **C** This is the length of the cutting edge or if it's a groove tool the groove tool width.

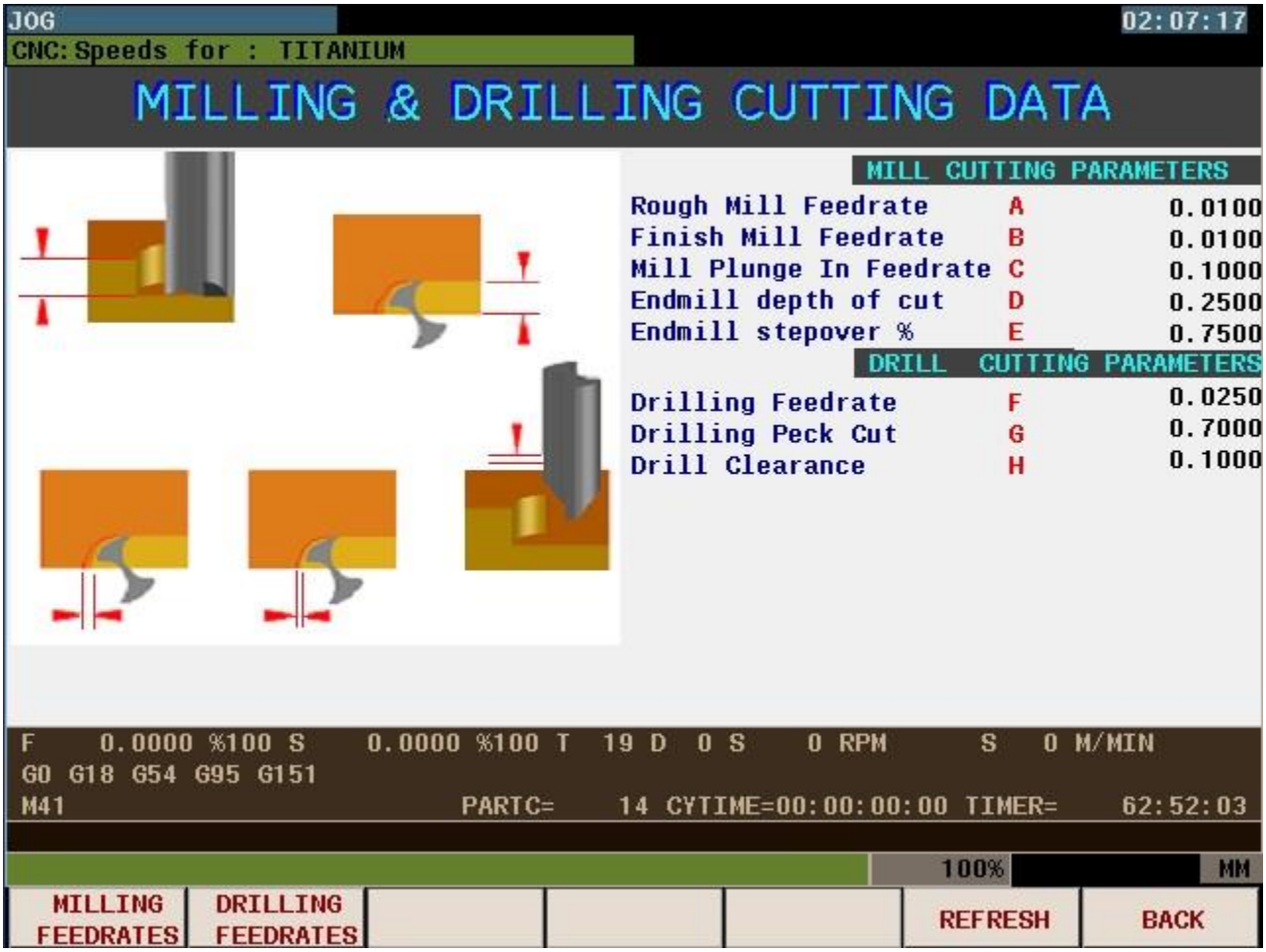
F4 **TOOL RADIUS**. **D** The tool radiusa is critical to the correct operation of the machine, if the radius is wrong you will see steps and lips in the finished rings.

F5 **TOOL TIP DIRECTION.** The cycles need to know which way the tool is orientated, use the graphic at the bottom right to select the correct orientation. All Form Tools should use 6 and the Z axis offset is to be set in the center of the tool.

F6 **DIFFERENT TOOL**. Live spindle speeds for different operations & materials.

F7 **BACK**. This key returns to the previous screen.

F3 MILLING FEED DATA (Cutting Technology)



There are 4 selections available:

F1 **MILLING FEEDRATES.** Feed rates for the different milling operations.

Rough Milling Feed rate

Finish milling feed rate

Mill infeed into material feed rate

Depth of cut when roughing or when slotting deep channels & grooves.

% of stepover when roughing pockets.

F2 **DRILLING FEEDRATES.** Feed rates for the different drilling operations.

F3 **NO SELECTION.**

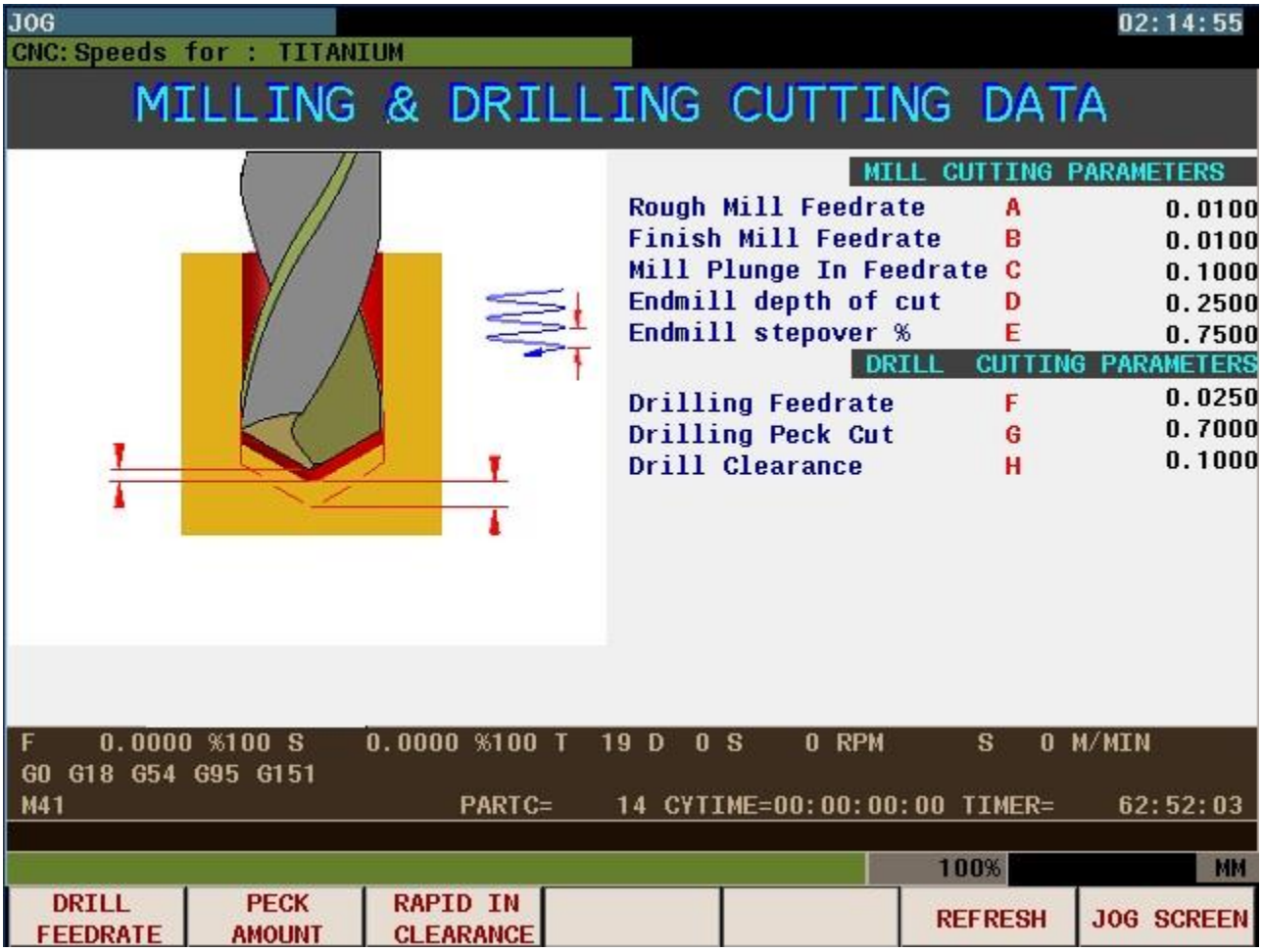
F4 **NO SELECTION.**

F5 **NO SELECTION.**

F6 **REFRESH. .** This reloads the values in memory so that they can be viewed when materials are changed.

F7 **BACK**. This key returns to the previous screen.

F3 DRILLING FEED DATA (Cutting Technology)



A separate graphic is displayed for the live tool drilling data.

There are 5 selections available:

F1 **DRILL FEEDRATES.** Feed rates for the different drilling operations.

F2 **PECK AMOUNT**. This is the amount of each advance when deep hole drilling or drilling tough materials.

F3 **RAPID IN CLEARANCE.** This is the amount the drill leaves for clearance when it rapids into the hole.

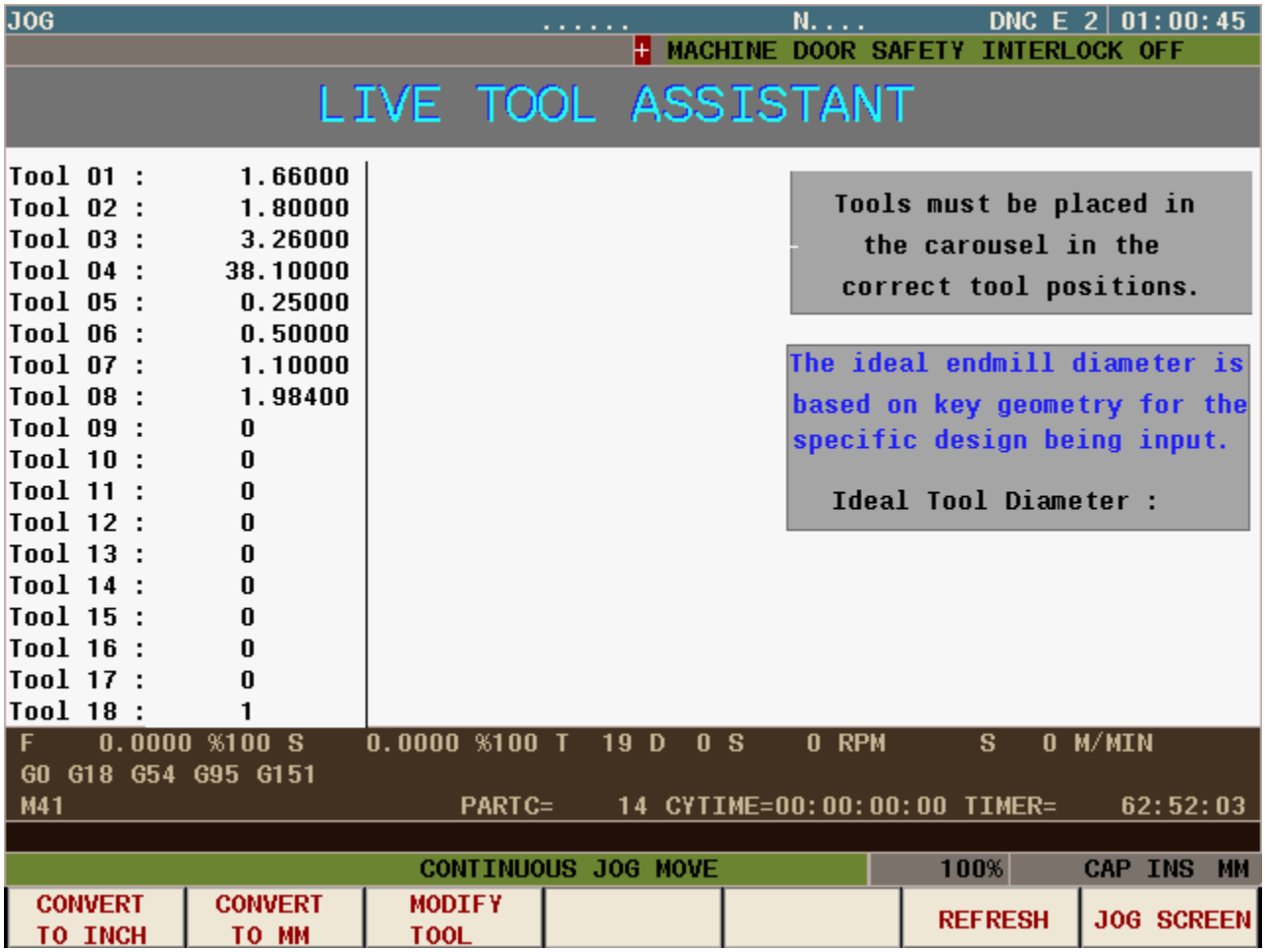
F4 **NO SELECTION.**

F5 **NO SELECTION.**

F6 **REFRESH..** This reloads the values in memory so that they can be viewed when materials are changed.

F7 **JOG SCREEN.** This key returns to the JOG screen.

F4 MILLING TOOL DATA (Cutting Technology)



This page provides information on the live tools in the carousel.

There are 5 selections available:

F1 **CONVERT TO INCH.** All values for the machine are in mm's this allows you to see the diameters in inches. (THIS DOES NOT CHANGE THE VALUES IN THE TOOL TABLE.)

F2 **CONVERT TO mm's**. This reconverts the values to mm's.

F3 **MODIFY TOOL.** This provide a way to change the diameters without using the tool table.

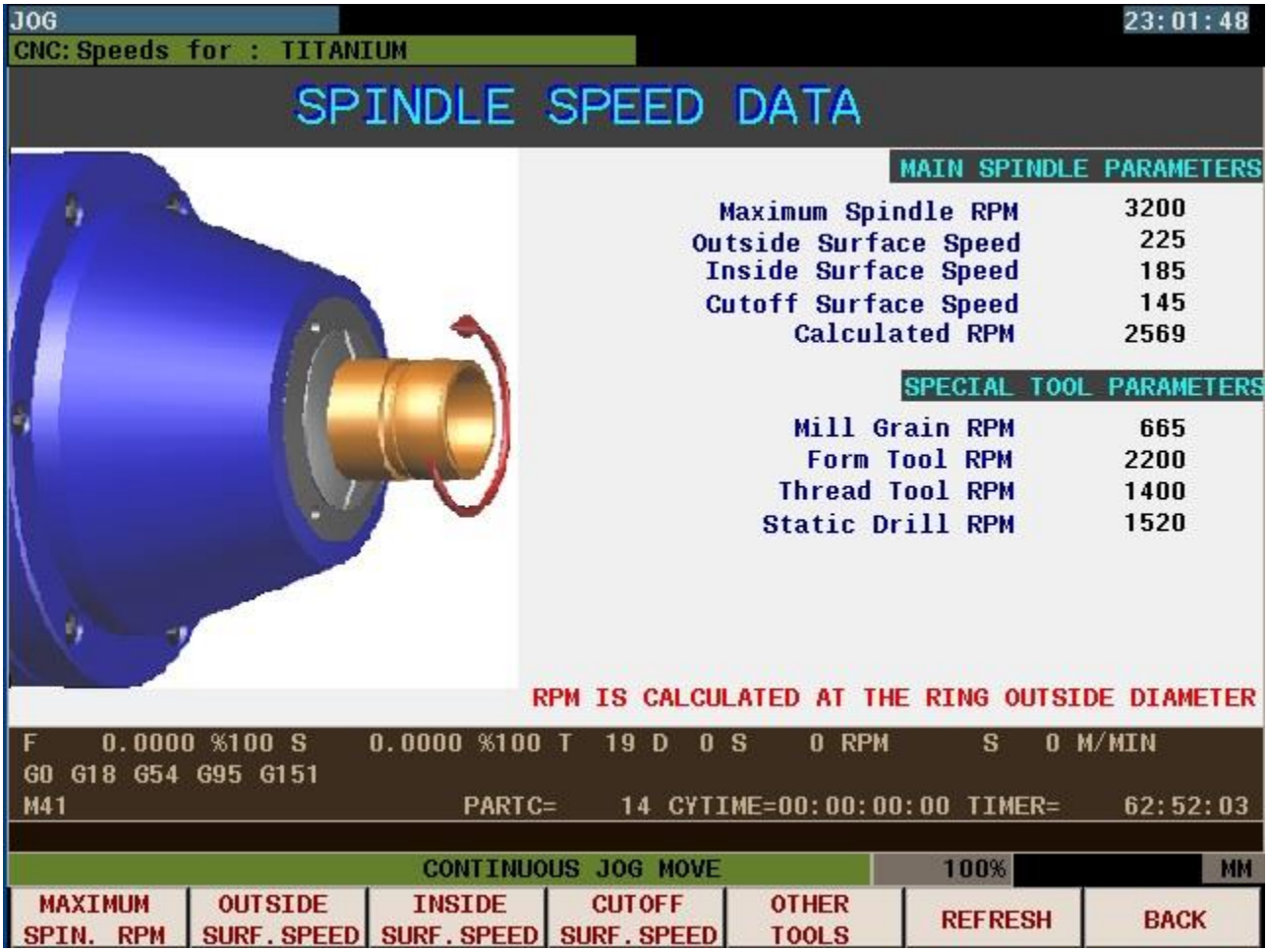
F4 **NO SELECTION.**

F5 **NO SELECTION.**

F6 **REFRESH.** This reloads the values in memory so that they can be viewed when adjustments are made.

F7 **JOG SCREEN.** This key returns to the JOG screen.

F5 TURNING RPM (Cutting Technology)



The software controlling the spindle speeds varies the RPM depending on the diameter being machined.

For more information please read the Operator User Manual under G code G96.

There are 7 selections available:

F1 **MAXIMUM SURFACE SPEED.** The value is the highest RPM the spindle will reach when calculating the speed.

F2 **OUTSIDE SURFACE SPEED**. This is the speed value for the outside profile machining.

F3 **INSIDE SURFACE SPEED.** This is the speed value for the inside bore profile machining.

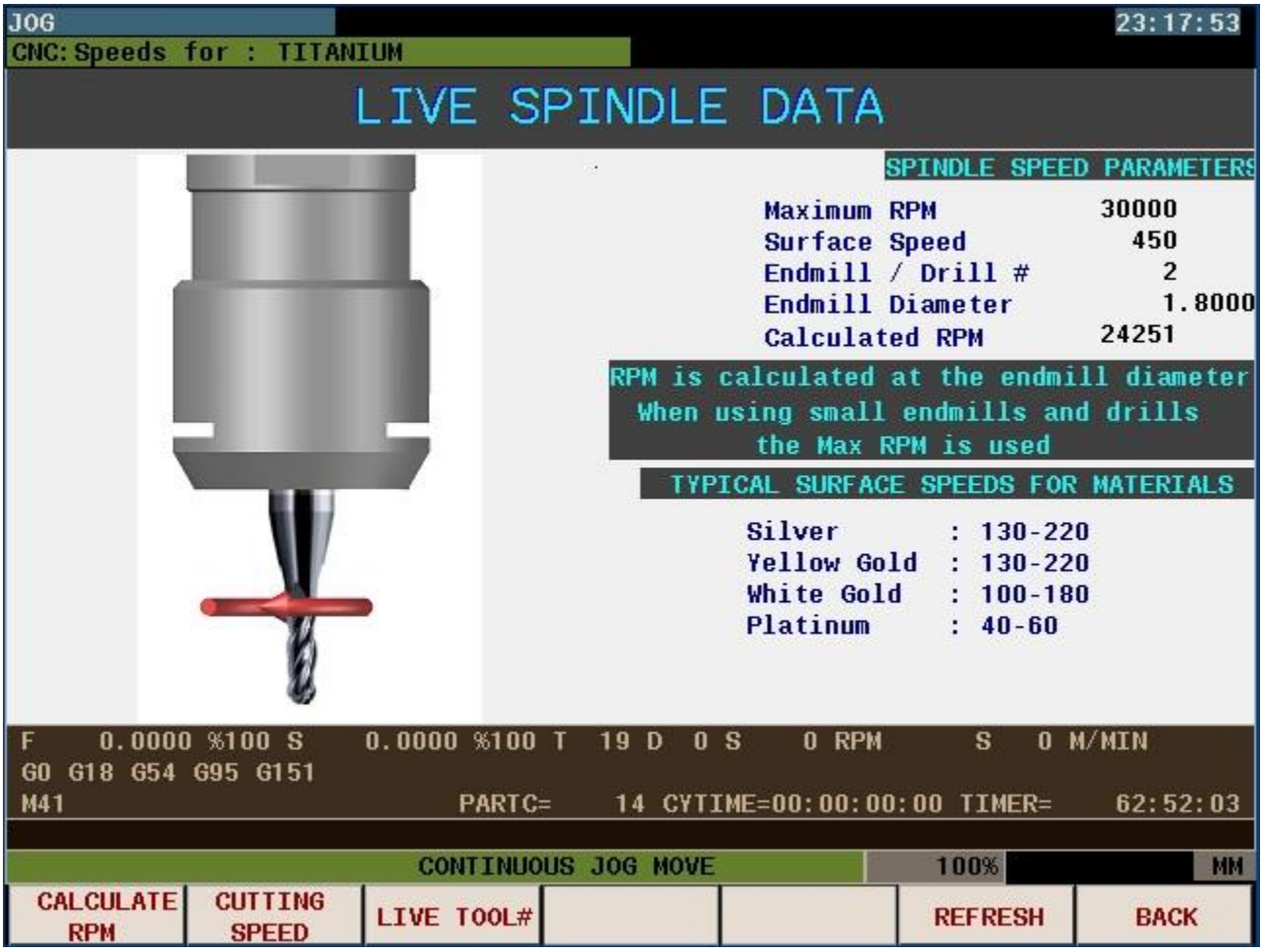
F4 **CUTOFF SURFACE SPEED.** This is the speed value for the cutoff tool operation.

F5 **OTHER TOOLS.** This selection allows input for the 4 special operations that use direct RPM input.

F6 **REFRESH.** This reloads the values in memory so that they can be viewed when materials are changed.

F7 **BACK.** This key returns to the previous screen.

F6 MILLING RPM (Cutting Technology)



There are 5 selections available:

F1 **CALCULATE RPM.** This will use the surface speed and the diameter of the tool selected to calculate the live spindle RPM.

F2 **CUTTING SPEED**. Enter the cutting speed for the material.

F3 **LIVE TOOL #.** Enter the live tool # to review the RPM

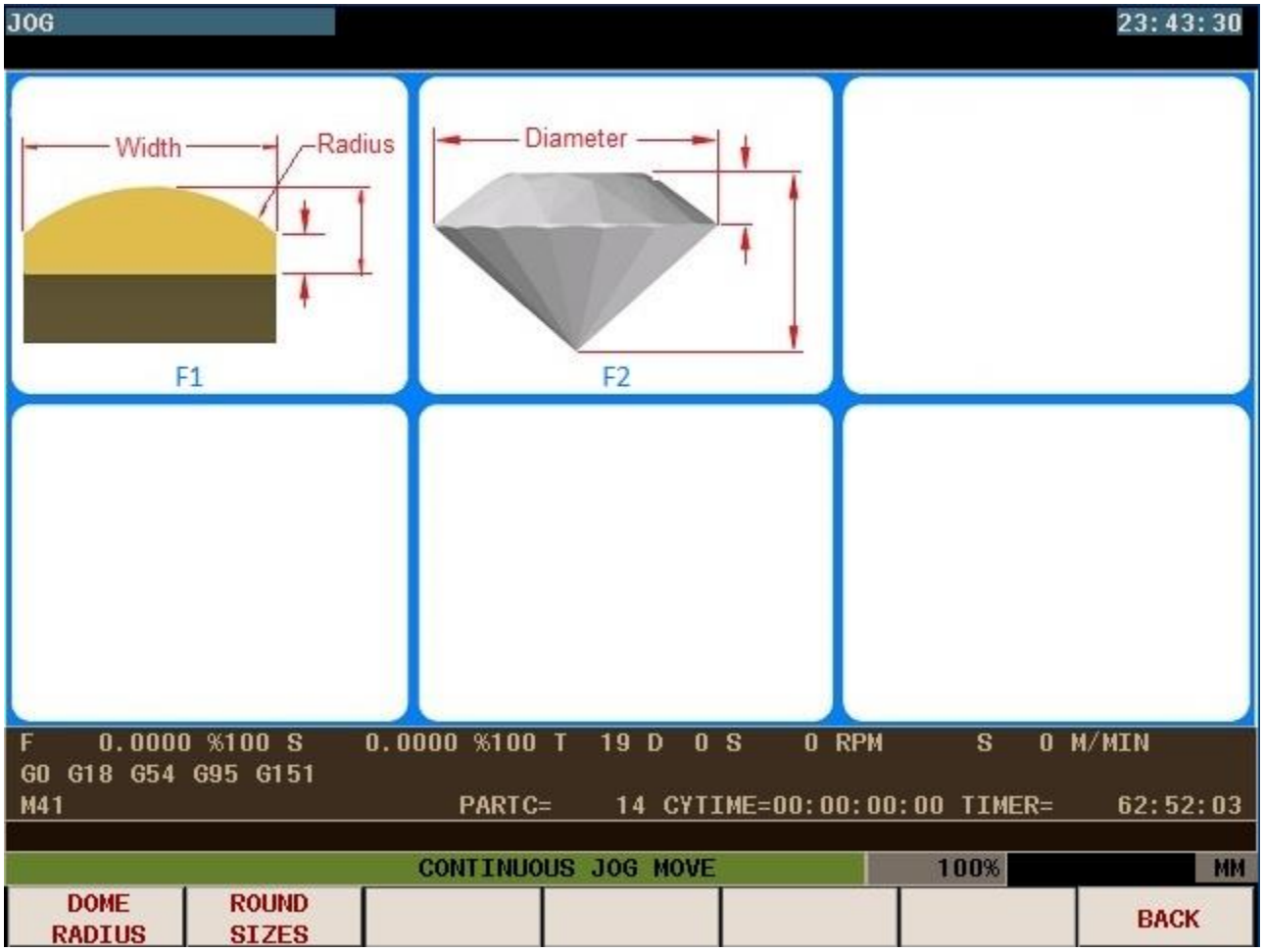
F4 **NO SELECTION.**

F5 **OTHER TOOLS.** This selection allows input for the 4 special operations that use direct RPM input.

F6 **REFRESH.** This reloads the values in memory so that they can be viewed when materials are changed.

F7 **BACK.** This key returns to the previous screen.

F3 Choice Screen (Interface/Calculator)



This section provides calculators to develop information for rings that you may need to make.

F1 **DOME RADIUS.** This calculator allows you to change the radii and width to calculate height

F2 **ROUND SIZES**. This calculates the different values of a round brilliant stone and other values.

F3 **NO SELECTION.**

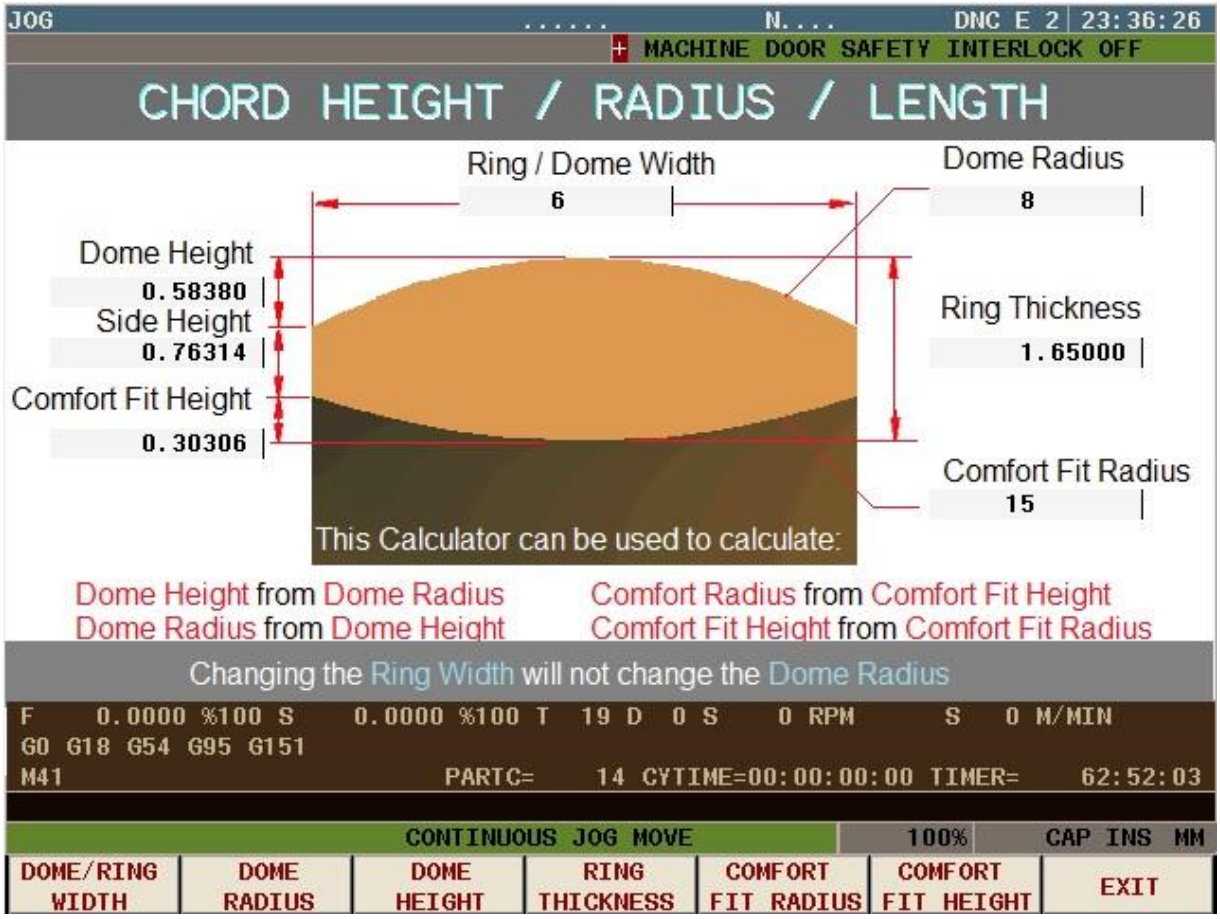
F4 **NO SELECTION.**

F5 **NO SELECTION.**

F6 **NO SELECTION.**

F7 **BACK.** This key returns to the previous screen.

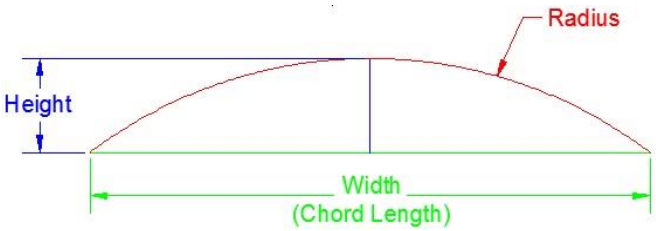
F1 DOME RADIUS (Interface/Calculator)



A simple profile with an outer & inner radius has quite a lot of math associated with the shape.

An arc has 3 features:

- **Width**
- **Height**
- **Radius**



There are 7 selections available:

Change any 1 value and all the other values change, This calculator assists in this calculation.

F1 **DOME/RING WIDTH.** Input the dome width, This keeps the dome radius and recalculates for the height.

F2 **DOME RADIUS**. Changing the dome radius will recalculate the dome height.

F3 **DOME HEIGHT.** Changing the dome Height recalculates the radius.

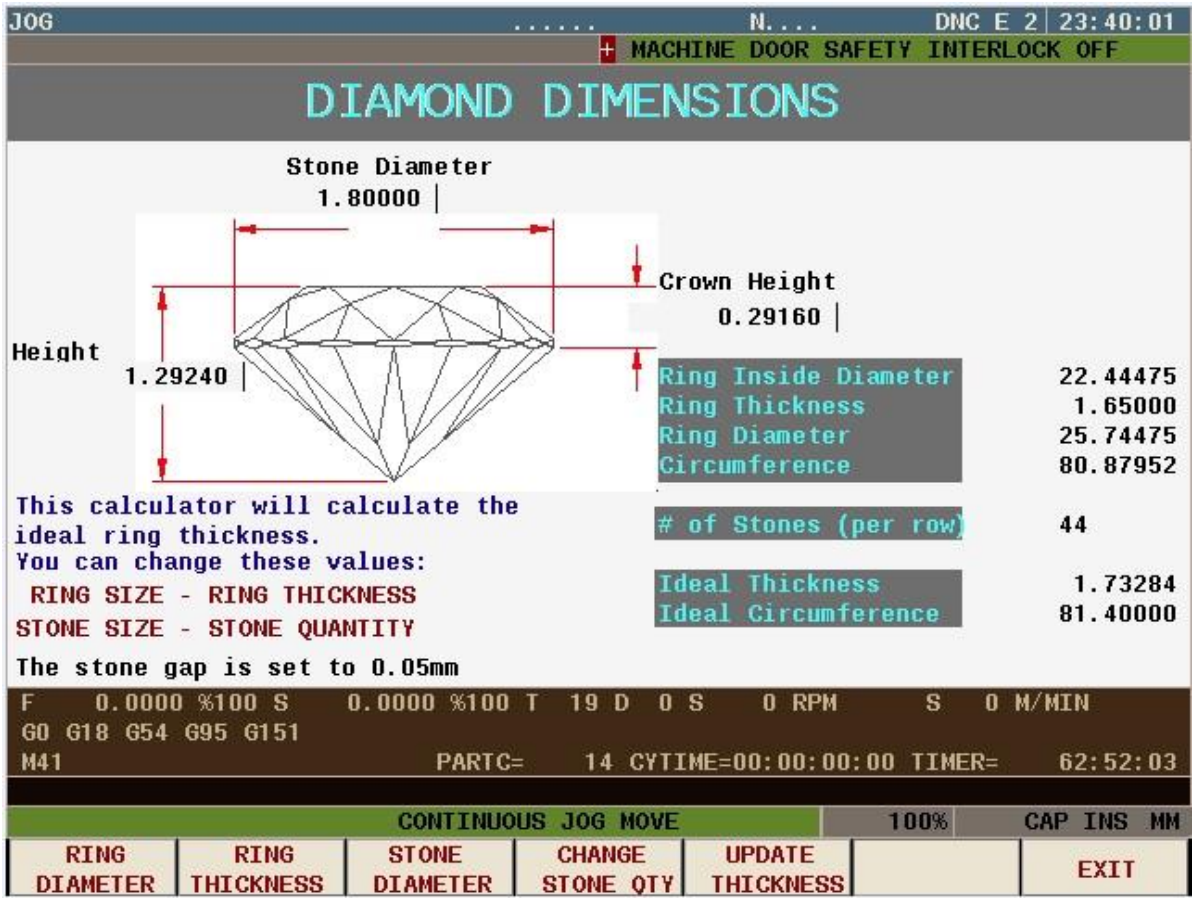
F4 **RING THICKNESS.** Change the ring thickness to match the ring you need to make.

F5 **COMFORT FIT RADIUS.** Changing the comfort fit radius will recalculate the arc height.

F6 **COMFORT FIT HEIGHT.** Changing the comfort fit Height recalculates the comfort fit radius.

F7 **EXIT.** This key returns to the JOG screen.

F2 ROUND SIZES (Interface/Calculator)



This calculator can help when eternity band values need to be calculated.

The parameters that control the diamond sizes are :

- P2227 = Stone gap value
- P2229 = Stone Height %
- P2230 = Crown Height %

When creating eternity bands the wall thickness is tied to the stone size and # of diamonds.

The ring circumference controls the number of stones, this value is based on the ring size + wall thickness x 2.

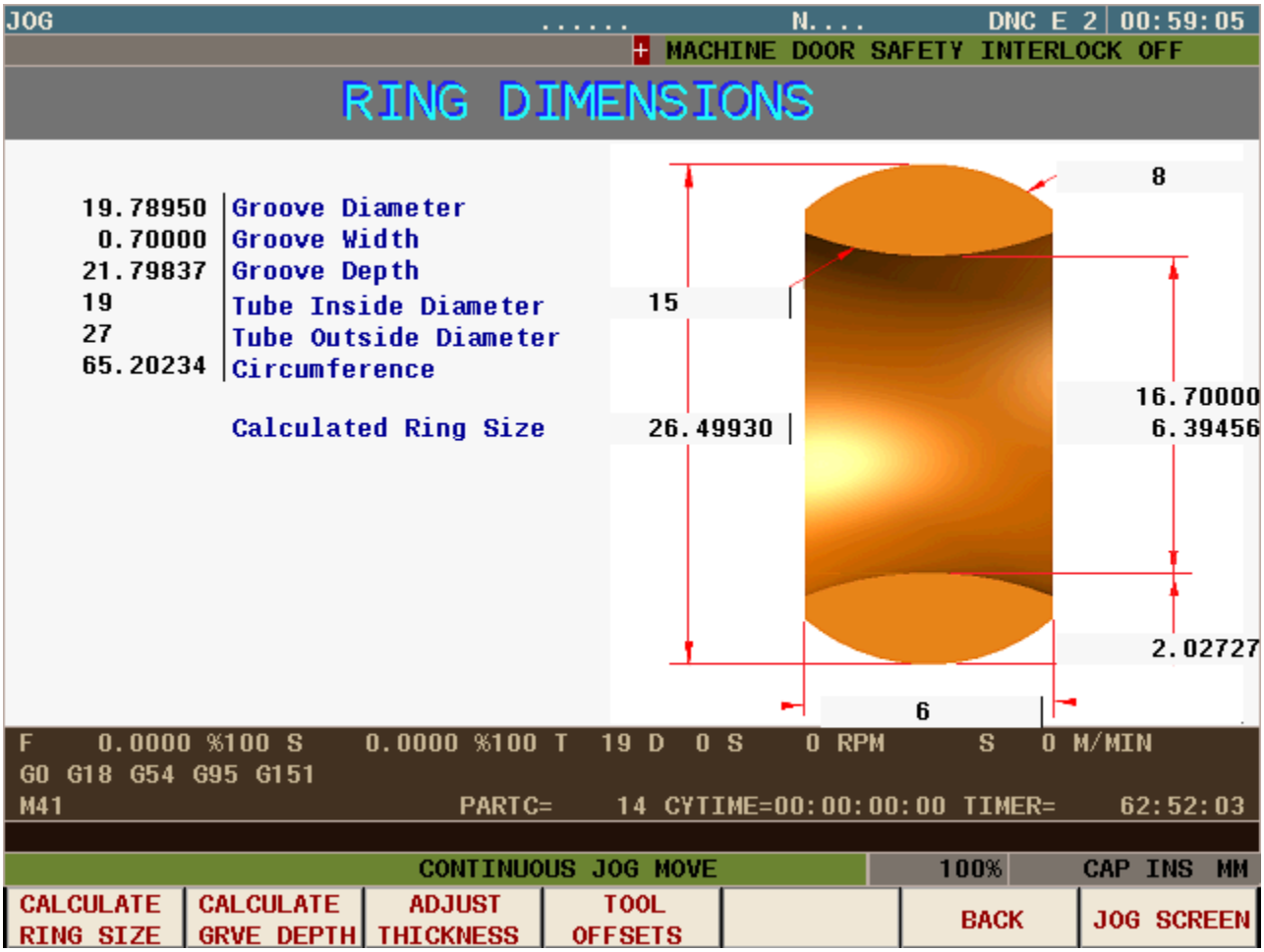
The wall thickness has to be greater than the stone height, and to keep the gap consistent the wall thickness needs to change.

The screen shows an **ideal thickness** this is based on the number of stones.

Changing the ring size will change the # of stones.

Changing the stone diameter will change the number of stones.

F4 RING INSPECTION



This page provides information on the rings you are making. The table shows relevant values.

Run a program in **SIMULATE** mode to load the values and use this page to check the ring dimensions.

There are 6 choices:

F1 **CALCULATE RING SIZE.** This allows you to enter a metric value and it calculates the specific ring size to use. This is useful on 2 piece rings where you need to produce a ring to a specific diameter.

F2 **CALCULATE GROOVE DEPTH.** grams.

F3 **ADJUST THICKNESS.** This sets the parameters so that barfeed operations cannot be performed.

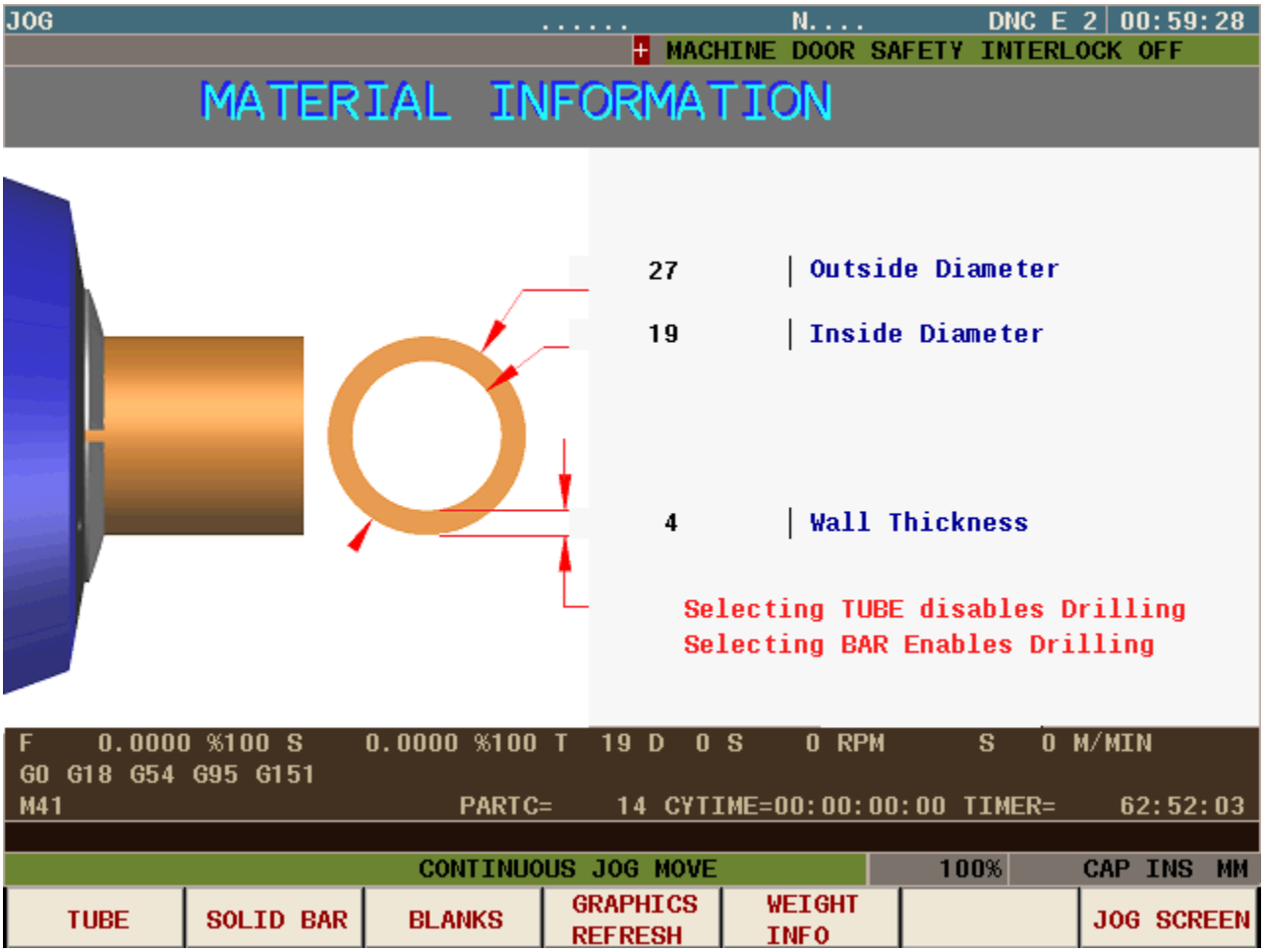
F4 **TOOL OFFSETS.** This keys opens up the tool offset page to allow changes to the tool offsets.

F5 **NO SELECTION.**

F6 **BACK.** This key returns to the previous screen.

F7 **JOG SCREEN.** This key returns to the JOG screen.

F5 MATERIAL INFORMATION



This page allows you to set the type of machining to be done.

There are 6 choices:

F1 **TUBE.** This button allows you to enter the tube size and wall thickness.

F2 **SOLID BAR.** This sets the parameters to activate the drilling cycles when creating new programs.

F3 **BLANKS.** This sets the parameters so that barfeed operations cannot be performed.

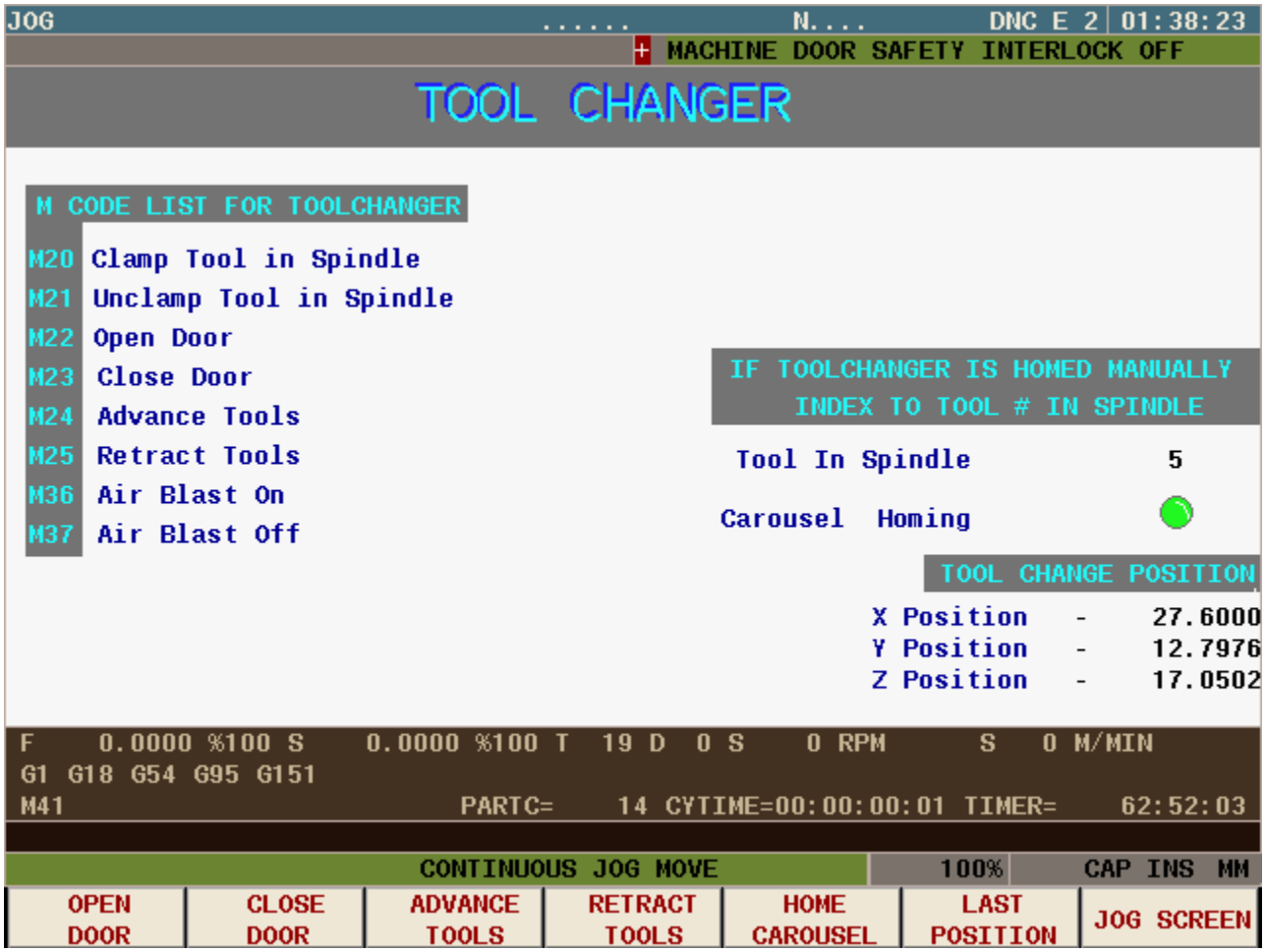
F4 **GRAPHICS REFRESH.** This uses the new tube and bar dimensions for the graphics screen.

F5 **WEIGHT INFO.** This section provides information on ring weights and material costs.
(This section is under development and new features will be available soon)

F6 **NO SELECTION.**

F7 **JOG SCREEN.** This key returns to the JOG screen.

F6 TOOLCHANGER



This page allows manual control of the tool changer mechanism.

It also lists the M codes related to the tool changer.

F1 **OPEN DOOR.** Press this button to open the tool changer door.

F2 **CLOSE DOOR**. Press this button to close the tool changer door.

F3 **ADVANCE TOOLS.** Press this button to advance the tool carousel.

F4 **RETRACT TOOLS.** Press this button to retract the tool carousel.

F5 **HOME CAROUSEL.** This button will home the carousel (Make sure to open the door first)

F6 **LAST POSITION.** This button will move the carousel to the tool position of the tool in the spindle.

F7 **JOG SCREEN.** This key returns to the JOG screen.

Each tool position has it's own Y-Z coordinates to provide alignment for the tool changer.

The values are store in subroutine (SUB 1515) depending on your configuration this subroutine may be in its own location or may be integrated into OEM Subroutines (P090000.PIT)