Training Center

Desktop Mill tutorial

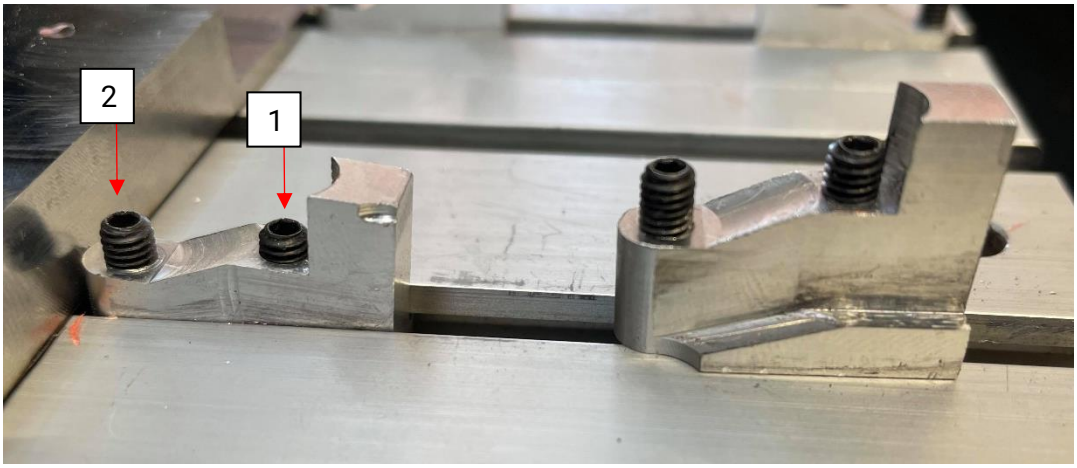
ECAM LYON_THAMMASAT UNIVERSITY

1 Table of contents

2 Key holder tutorial

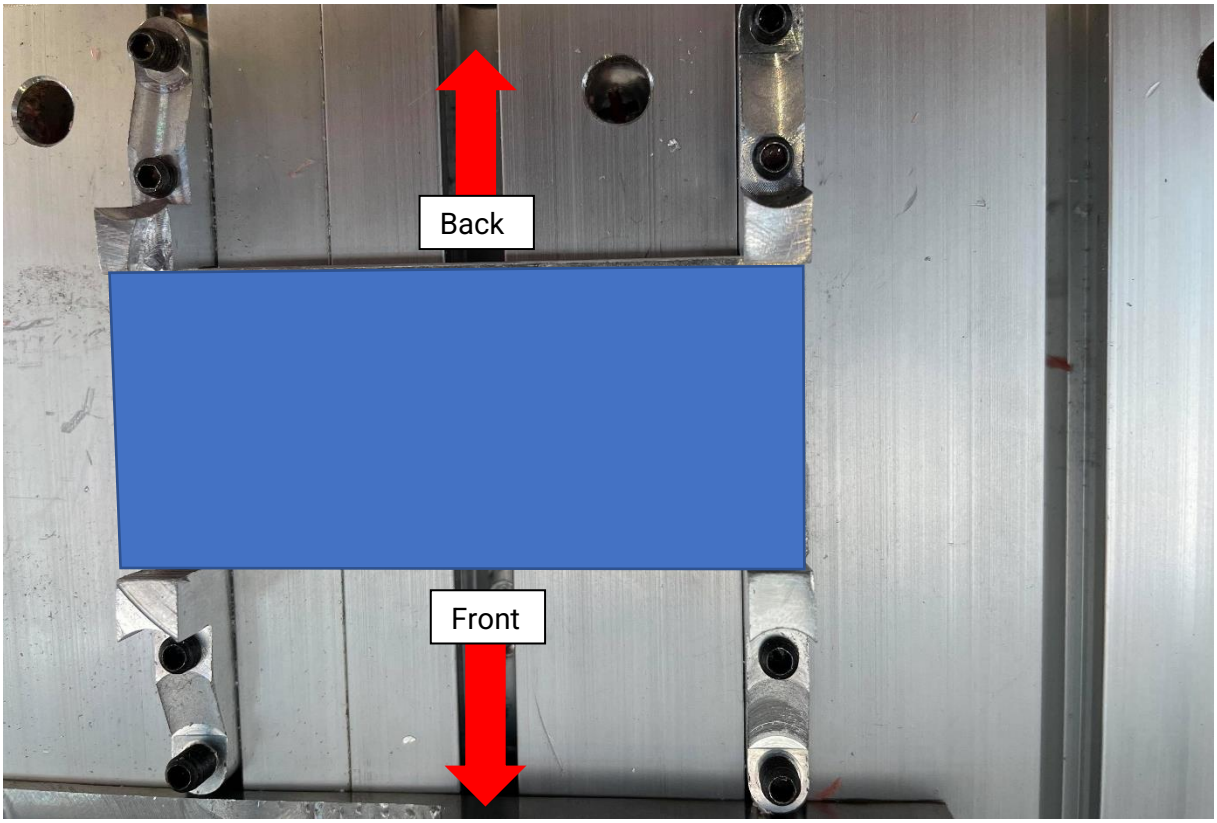
2.1 Setting the raw in place

1- setting the plate in place

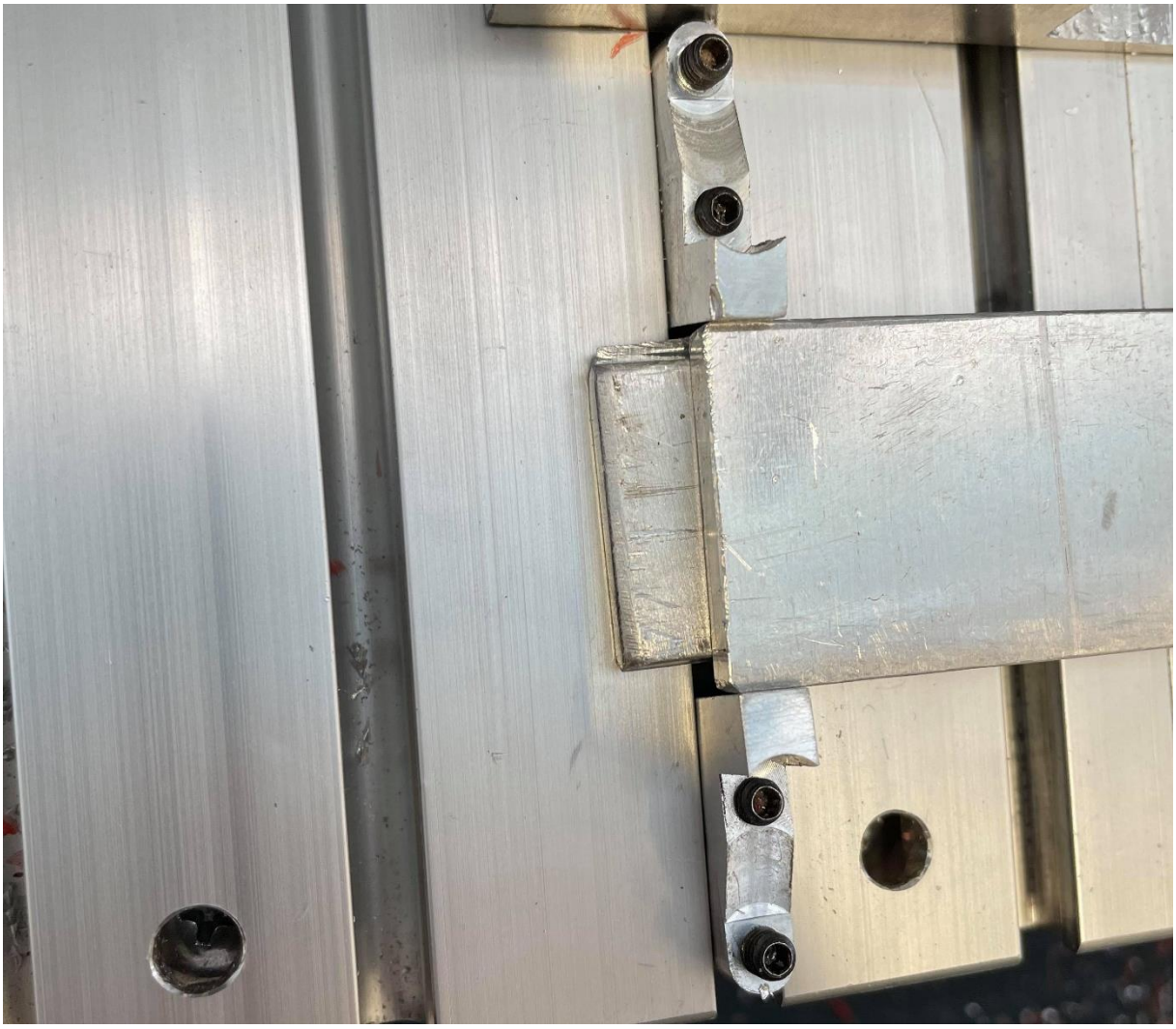


Slide them in the table grooves, then screw the screw #1 to set the flange in place then screw the screw #2 to allow it to swing.

Put the four flanges in place as shown bellow but don't tighten them yet. Put a martyr plate in place then the plate that you want to engrave.



In order to put your plate in place in the good way, you need to set it flush with the flanges on the left side.



You then need to tighten all the screws, beginning by the #1 screws then the #2 screws. Check if the plate is moving, if it is, you need to tighten it again. Gently hit it with a mallet to put it flat with the surface bellow.

2.2 Machine Power-Up

This section tells you how to power-on a new machine for the first time.

1. Press [POWER ON] until you see the Haas logo on the screen. After a self-test and boot sequence, the display shows the startup screen.

The startup screen gives basic instructions to start the machine. Press [CANCEL] to dismiss the screen.

Turn [EMERGENCY STOP] to the right to reset it.

2. Press [RESET] to clear the startup alarms. If you cannot clear an alarm, the machine may need service. Contact your Haas Factory Outlet (HFO) for assistance.

3. If your machine is enclosed, close the doors.

WARNING:

Before you do the next step, remember that automatic motion beginsimmediately when you press **[POWER UP]**. Make sure the motion path is clear. Stay away from the spindle, machine table, and tool changer.

1. Press **[POWER UP]**.

After the first **[POWER UP]**, the axes move toward their home positions. The axesthen move slowly until the machine finds the home switch for each axis. This establishes the machine home position.

2. Press any of the following:
a. **[CANCEL]** to dismiss the screen.
b. **[CYCLE START]** to run current program.
c. **[HANDLE JOG]** for manual operation.

2.3 Spindle Warm-Up

If your machine's spindle has been idle for more than (4) days, run the spindle warm-up program before you use the machine. This program brings the spindle up to speed slowly to distribute the lubrication and let the spindle reach a stable temperature.

Your machine includes a 20-minute warm-up program (O09220) in the program list. If you use the spindle at consistent high speeds, you should run this program every day. Go to "2.5 Loading the Program" to load it.

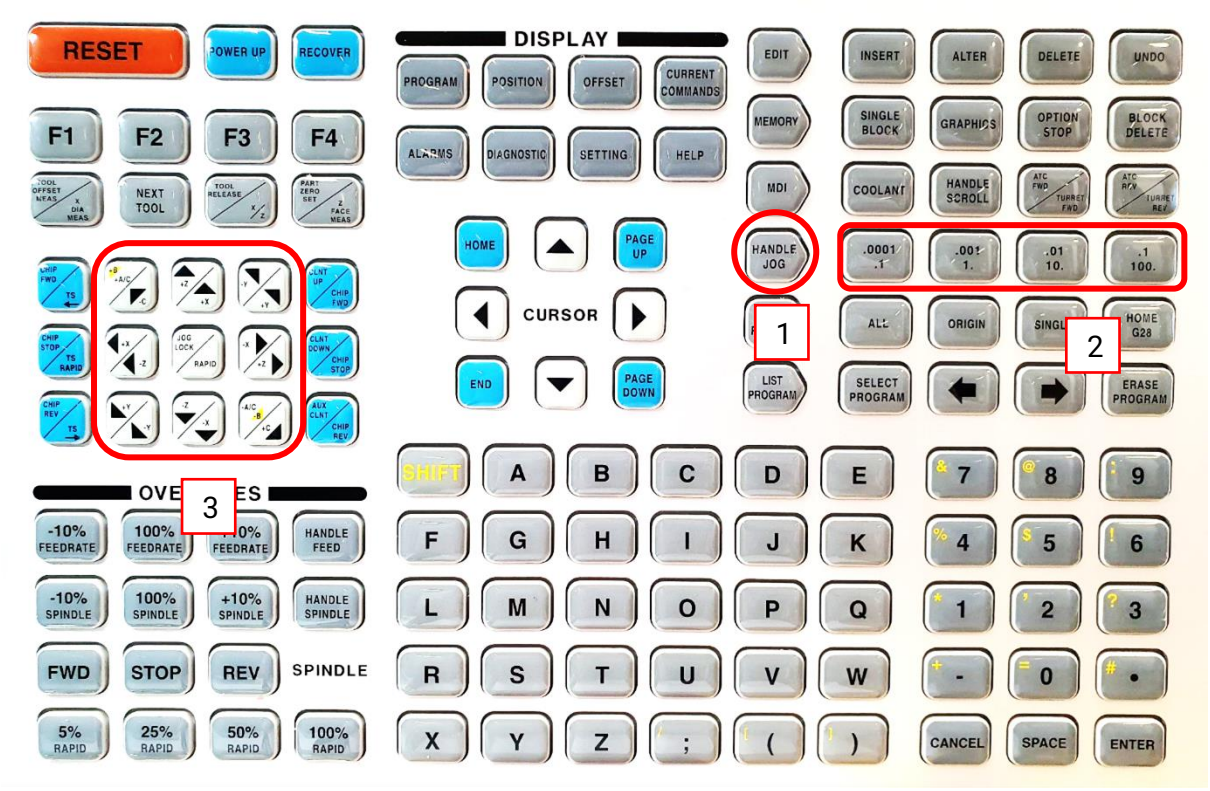
2.4 Define the coordinate system.

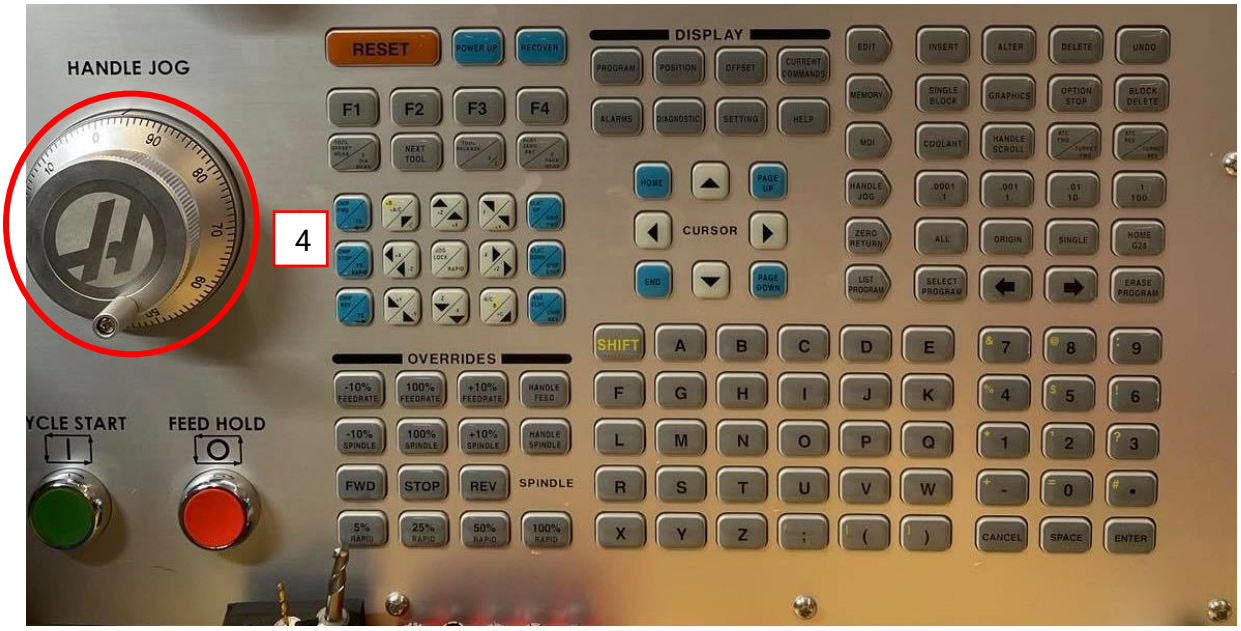
After doing the start-up procedure, in order to move the axis, you need to click the “Handle jog” button (1)

select the speed from “.0001” being 1 micrometre to “.1” being 1 millimetre (2), we recommend using the low speeds when approaching the piece to avoid damaging the workpiece.

Select the axis on which you would like to move with the corresponding buttons (3).

Use the “handle jog” (4) to move along the selected axis.





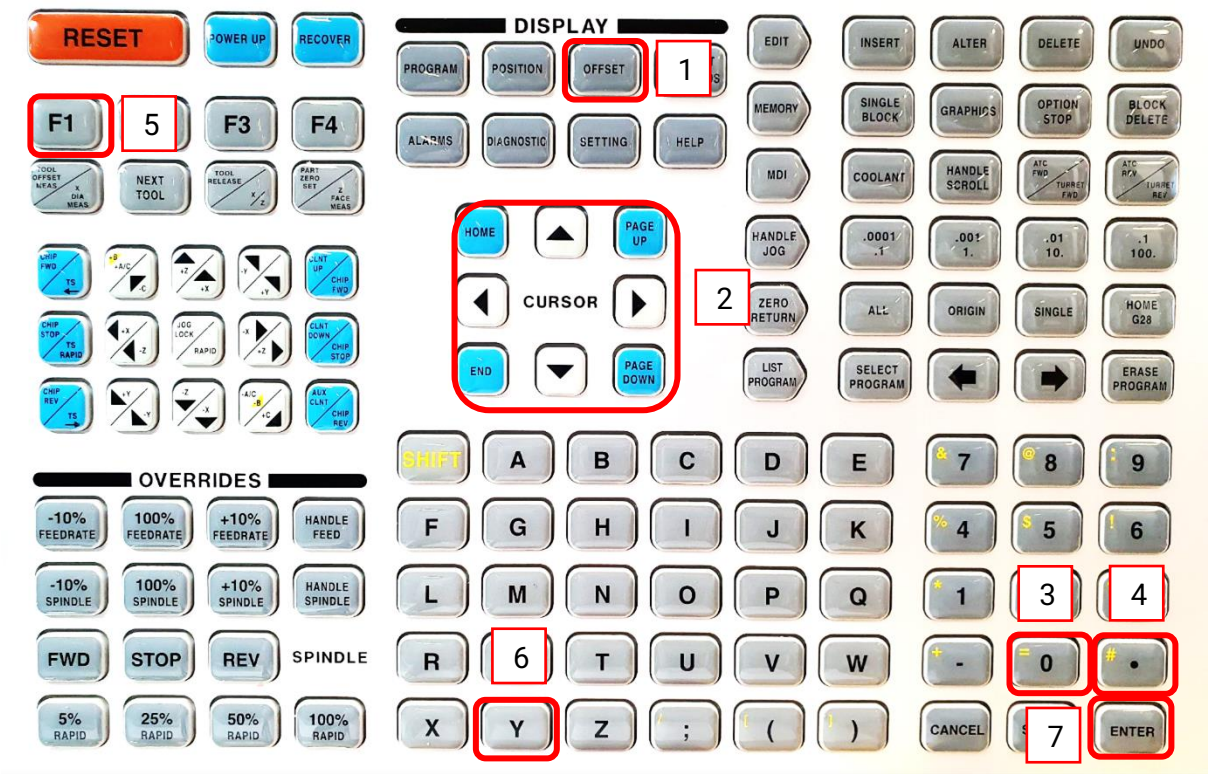
To define the coordinate system, you need to touch the left side of the workpiece with the tool as shown below by moving it as explained previously. Once your tool is in contact with the piece:

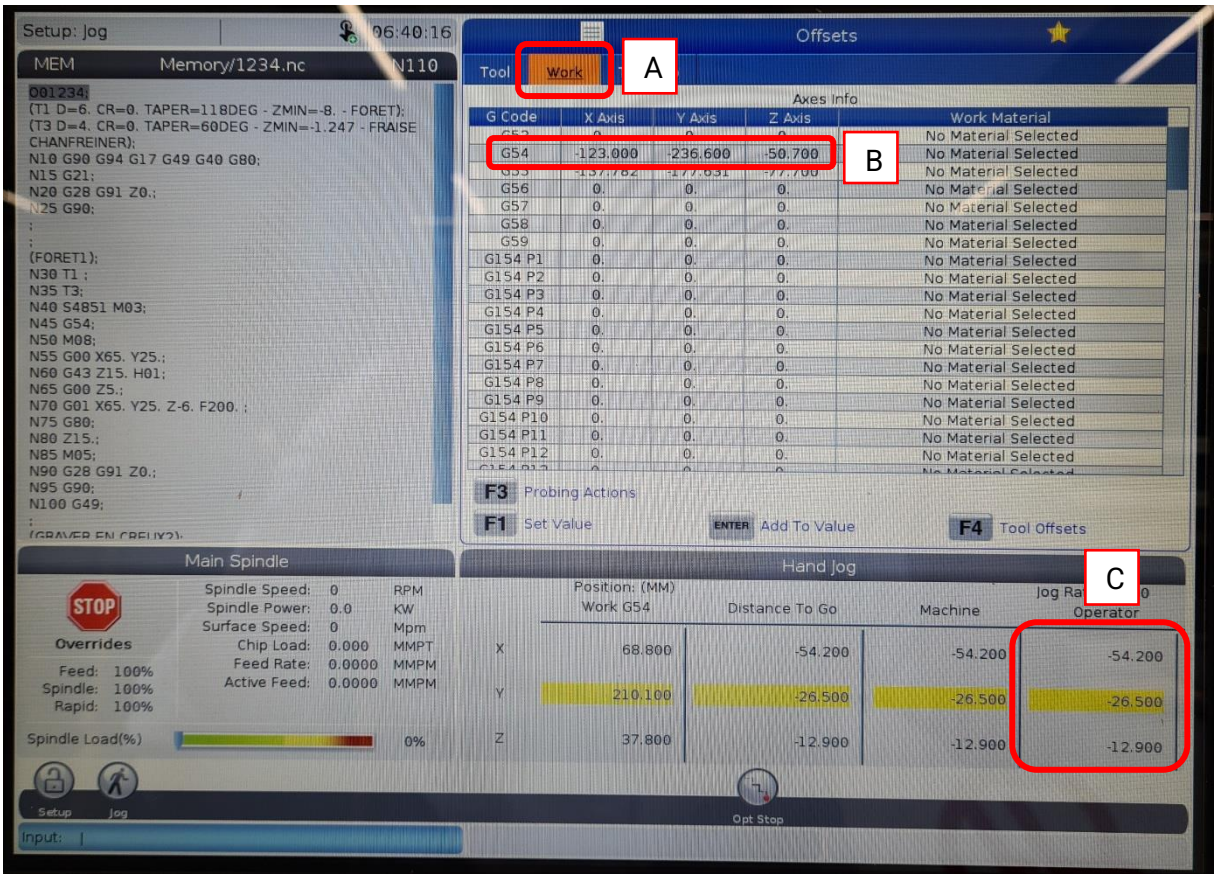
click the “offset” button (1) then click on the “Work” (A) tab on the touchscreen then go to the “G54” (B) line using the arrows (2) and set the “x” coordinate to 0 by clicking “0” (3) then “.” (4) then “F1” (5) and “Y” (6).

In the same tab, read the X coordinate in the right column (C) in the hand jog section of the screen and write it in the G54 line in the “X axis” column. Press “Enter” (7) then “Y” (6).

Type “-3.” And “Enter”, 3 being the radius of the drill bit.

Do the same thing for the Y axis by touching the front of the piece and on the Z axis by touching the upper part of the piece but don’t subtract the radius for the upper surface.




Setup: Jog
06:40:16
MEM
Memory/1234.nc
N110
O01234;
(T1 D=6. CR=0. TAPER=118DEG - ZMIN=-8. - FORET);
(T3 D=4. CR=0. TAPER=60DEG - ZMIN=-1.247 - FRAISE CHANFREINER);
N10 G90 G94 G17 G49 G40 G80;
N15 G21;
N20 G28 G91 Z0.;
N25 G90;
;
;
(FORET1);
N30 T1 ;
N35 T3;
N40 S4851 M03;
N45 G54;
N50 M08;
N55 G00 X65. Y25.;
N60 G43 Z15. H01;
N65 G00 Z5.;
N70 G01 X65. Y25. Z-6. F200. ;
N75 G80;
N80 Z15.;
N85 M05;
N90 G28 G91 Z0.;
N95 G90;
N100 G49;
;
Offsets
Tool
Work
A
Axes Info
G Code
X Axis
Y Axis
Z Axis
Work Material
G54
-123.000
-236.600
-50.700
No Material Selected
B
G56
G57
G58
G59
G154 P1
G154 P2
G154 P3
G154 P4
G154 P5
G154 P6
G154 P7
G154 P8
G154 P9
G154 P10
G154 P11
G154 P12
F3 Probing Actions
F1 Set Value
ENTER Add To Value
F4 Tool Offsets
Main Spindle
STOP
Overrides
Feed: 100%
Spindle: 100%
Rapid: 100%
Spindle Speed: 0 RPM
Spindle Power: 0.0 KW
Surface Speed: 0 Mpm
Chip Load: 0.000 MMPT
Feed Rate: 0.0000 MMPM
Active Feed: 0.0000 MMPM
Spindle Load(%) 0%
Hand Jog
C
Position: (MM)
Work G54
Distance To Go
Machine
Operator
X 68.800 -54.200 -54.200 -54.200
Y 210.100 -26.500 -26.500 -26.500
Z 37.800 -12.900 -12.900 -12.900
Setup
Jog
Opt Stop
Input:

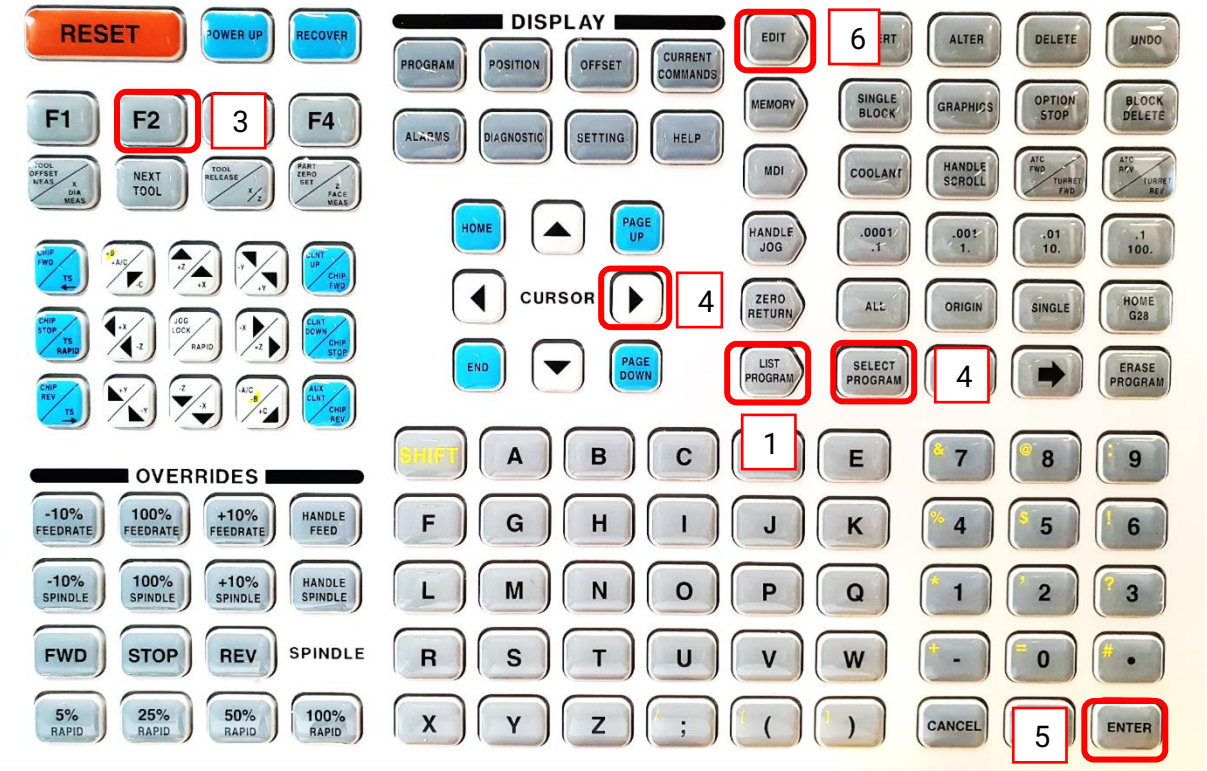
2.5 Loading the program

Insert the USB key to the right of the control panel.

To get the program from the USB key, press “list program” (1) then go in the “USB” menu (A) and use the arrows to choose your program. Press “F2” (2) to copy it then paste it in the memory of the machine by pressing the right arrow (3) on “memory” then “enter” (2). You can now go back to the “Memory” tab.

If the program is already in the memory, you can press the “list program” (1) then go to your program using the arrows the press “select program” (5) button.

If you need to, you can click the “Edit” (6) button to see and modify your program.



2.6 Launching the program

For the first piece, we recommend slowing the speed to 5-25% by pressing the “5% RAPID” and the “25% RAPID” buttons (1) to have the time to see if anything goes wrong. You can also press the “Handle feed” (3) button then use the “Handle Jog” (4) to control the speed. Click the “Memory” (5) button and launch the program by pressing “Start cycle” (6).

For each operation, we recommend making sure that the spindle gets up to speed by using the handle jog to set “rapid” to “0” and pressing the “100% spindle” button. Once it’s spinning at the right speed, you can adjust the “Rapid” line back to the working speed.

Once the Drilling operation is done, the program will stop for you to change the tool. Once the tool is changed, press the “Reset” (1) button then set the z coordinate following the “2.4 define

the coordinate system" procedure. If you don't want to damage the surface of the workpiece you can slide a piece of paper between the tool and the piece and slide it a little bit. When you can't move the paper anymore, you know that the tool is touching the workpiece. Enter your new Z value then enter "0." then "F1" then "Y" then type "-0.3" then "Enter" to subtract the thickness of the piece of paper.

Once this operation is done, press the "Edit" (2) button and go down to the line "N105 M01" under "(GRAVER EN CREUX 2)" using the arrows. Press the "Memory" button, on the left part of the screen, the line "N105 M01" should be highlighted, indicating that the program will start from this line. You can now start the program by pressing "Start cycle".

The tool will go up then stop. You will have to press "Start cycle" once again. Let the spindle come up to speed as you did before, doing the same procedure.