HOW TO USE FANUC MIRROR FUNCTION

(Tested for Hyundai-Kia VX400 VX500 VX650 with Fanuc 0iMC)

- MIRROR THRU MDI COMMAND

1. Call the proper work offset (G54 etc.).

2. Position the X Y at the mirror center line, record this value (Point A).

3. Go to MDI mode -> Offset Hard Key -> Offset Soft Key -> Page Down, change Mirror input from 0 to I on X (or Y).

4. Proceed to run the program, it shall be mirrored. Raise the work offset Z value to verify for safety in advance.

5. After finish the operation, move spindle back to Point A,

6. Go to MDI mode -> Offset Hard Key -> Offset Soft Key -> Page Down, change Mirror input from 1 to 0 or X (or Y).

7. Bear in mind before turn on or off the mirror switch, the position of the spindle is important, it should be the on the centerline.

8. If mess up the coordinate due to improper procedure, you can go to MDI mode change mirror switch to 0 and power down power up the machine, this should reset machine back to original state prior to the changing of mirror switch.

9. If you have multiple mirrored parties/cavities need to machine, you still can use G52 to shift the program in different locations. Just remember once mirrored, the X+ or Y+ are in the revered directions.

Disclaimer: Use producers at your own risks, run CNC with caution, verify in air before cutting, we are not responsible for any damages.
• MIRROR THRU NC CODE

%
O0006
G49G90G80G0G40 ____________ Safety Block
T51M6 _____________________ Tool Call
G0G54.1P1X0Y0 ____________ Go to mirror point
/G51.1X0 _________________ Turn on Mirror around X0 line
G43H51Z1. ________________ Call Tool Length Offset
M1 _____________________ Optional stop
G0X-6.Y-3. ________________ Positioning
G05.1Q1 ________________ Turn on AICC
G120F200. ________________ Motion
G41D51G1X-2. ____________ Motion with Cutter Comp On
Y3. ________________ Motion
X-6. ________________ Motion
Y-3. ________________ Motion
G0Z1. ________________ Retract
G40 _____________________ Cutter Comp Off
G05.1Q0 ________________ AICC Off
G0X0Y0 ________________ Go back to mirror point
/G50.1X0 ________________ Cancel Mirror Function
M1 _____________________ Optional Stop
G91G28Z0 ________________ Retract to Machine Home
M30 _____________________ End
%

Note:

1. If the program is interrupted in the middle, make sure G40 is called manually in MDI or in program when restart.