MACRO
SHOP
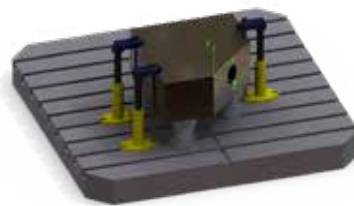
OFFSET RECALCULATION

FANUC 0/16/18/21/31/32

Rev. 4

FIELD OF APPLICATION

The macro-instruction recalculates the offset in relation to the centre of the table, following the rotation of the rotary table. This macro-instruction can be used to create a new rotated or rotated and shifted offset from any basic one by setting the table rotation degrees. The only thing to be done in the macro is to enter the position of the table centre in the machine co-ordinates in parameters #127, #128 and #129. The macro can be used in two ways in order to meet various requirements. The first mode creates a new offset recalculated from the table centre, which is typical of machining operations carried out on boring machines where you wish to have new offsets but recalculated from the initial one, so that you can work independently on all the offsets during the finishing phases. A second case of use involves temporarily moving the active offset using the G52 function to reach the new position of the offset after the table rotation. At the end of the machining, simply deactivate the G52 function to return to the offset initially set.

CYCLE DESCRIPTION

The macro-instruction will be called using the G65 function, preceded by a call to the offset from which you wish to start.

The macro can be used in two ways:

CASE 1: CREATING A NEW OFFSET

G54 (OFFSET EXAMPLE)
G65 P8001 **B__ M__** X__ Y__ Z__ S__

PARAMETER DESCRIPTION:

G65P8001
P8001 is the program number with which the macro-instruction is provided. If the O8001 program is an unavailable program number, change the program number of the macro-instruction, then the P parameter must indicate the new program number. If, for example, the macro-instruction is renumbered to O1500, the call will become G65P1500.

B_
Position of the table to be calculated as the new rotated offset. Enter both positive and negative values. Example B15, B270, B-35.

G65 P8001

B_

Made in Italy

M_

The letter M indicates the offset to be generated. Follow the scheme below:

M1 corresponds to the offset G54
M2 corresponds to the offset G55
M3 corresponds to the offset G56
M4 corresponds to the offset G57
M5 corresponds to the offset G58
M6 corresponds to the offset G59

In the case of a machine equipped with additional offsets
M101 corresponds to G54.1 P1
M102 corresponds to G54.1 P2
etc.

X_Y_Z_

The X, Y and Z coordinates are used to indicate any translation values of the offset. This is used when the translation at the offset should be absorbed directly. The translation can be defined with non-rotated coordinates or with already rotated coordinates. The behaviour will be defined using the S parameter.

S_

The S parameter is used to indicate the possible sequence for the offset translation. The X_Y_ and Z_ translation values will be interpreted by the macro according to the following convention:
- S0: the macro-instruction firstly rotates the offset and then translates the values indicated by X, Y and Z;
- S1: the macro-instruction firstly translates the values indicated by X, Y and Z and then rotates them around the table centre by the angle indicated by the B parameter.

C_

The C parameter defines the table position of the initial offset when it is different from zero. If it is not programmed, the table offset is considered to be zero degrees when calculating the rotations. (example B0)

Application note
If no translation of the origin is desired, the values of parameters X, Y, Z and S can be omitted.

CASE 2: OFFSET TRANSLATION

This macro performs the offset translation using the G52 function to reset the active offset to the position calculated after a table rotation. Then at the end of the machining, always enter G52 to deactivate it with G52X0Y0Z0. If it is necessary to use G52 within the program for any offset translations, it will also be possible in this case to set the translation co-ordinates using the X_Y_Z_ parameters. Also in this case, the S parameter will define whether to perform the translation before or after the rotation.

So the syntax will be:

G54
G65P8001B__

Where:

B_ Corresponds to the table's rotation degrees.

If you want to perform translations at the same time, the syntax will become:

G54
G65P8001B__X__Y__Z__S__

Where X, Y, Z and S have exactly the same meaning as seen for case no. 1.

C_

Indicate the C parameter with the value of the table position degrees for the start offset when the start offset is not zero degrees.

If, for example, the G54 origin corresponds to the table at position B180, set C180. See the example below:

In this case the parameter M is not given because the offset itself is to be transformed without creating a new one.

| Without C parameter	With C parameter
T1 M6 (DRILL 6.8)	T1 M6 (DRILL 6.8)
S6787 M3	S6787 M3
G54 (B180 OFFSET)	G54 (B180 OFFSET)
G65 P8001 B-70 (110-180)	**G65 P8001 B110 C180**
G0 B110	G0 B110
N240 X-46.448 Y174.139	N240 X-46.448 Y174.139
N250 Z64.959	N250 Z64.959
N260 G81 G98 Z21.922 R57.632 F1000	N260 G81 G98 Z21.922 R57.632 F1000
N270 G80	N270 G80
N280 G0 Z64.959	N280 G0 Z64.959
N290 Z227.291	N290 Z227.291

PROGRAMMNG

Here are 4 programming examples in which the origin recalculation macro is used both for generating new rotated or rotated-translated offsets and for moving the same active work offset.

The four examples all start from a G54 offset performed with a B0 table.
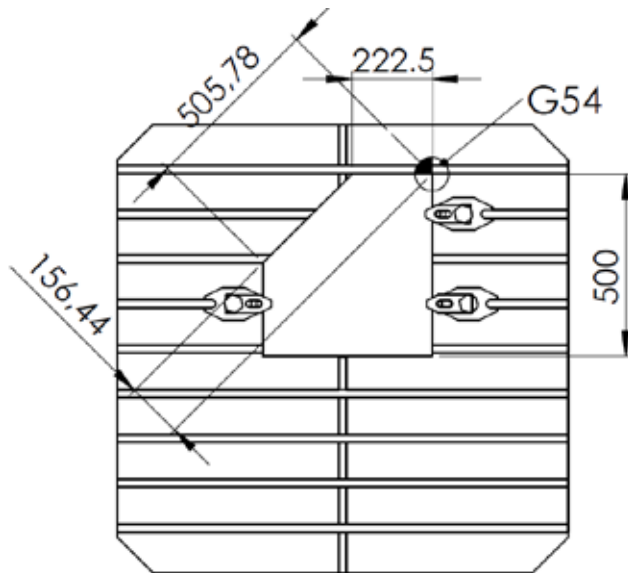
EXAMPLE N°1: creation of the G55 offset from G54 offset but with the table rotated by 45°.

```
G54
G65P8001B45M2
```


B45 TABLE
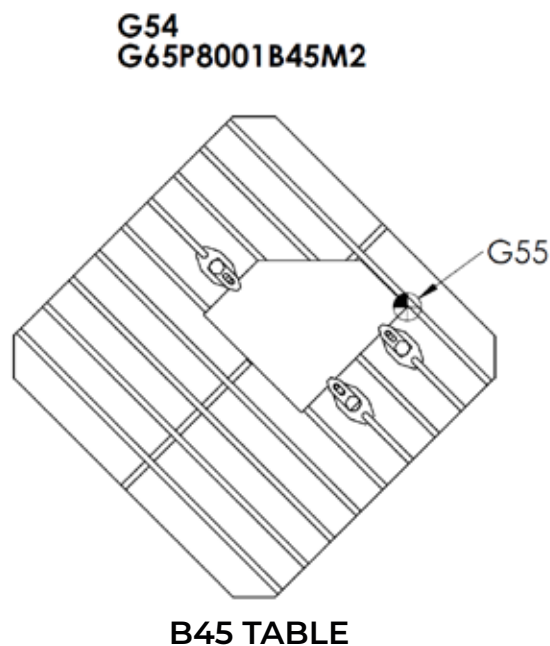
EXAMPLE N°2: Creation of a new G55 offset starting from a G54 offset but obtained by a translation of the offset along the X axis with values referring to the offset that has not yet been rotated. The S parameter was then set equal to 1.
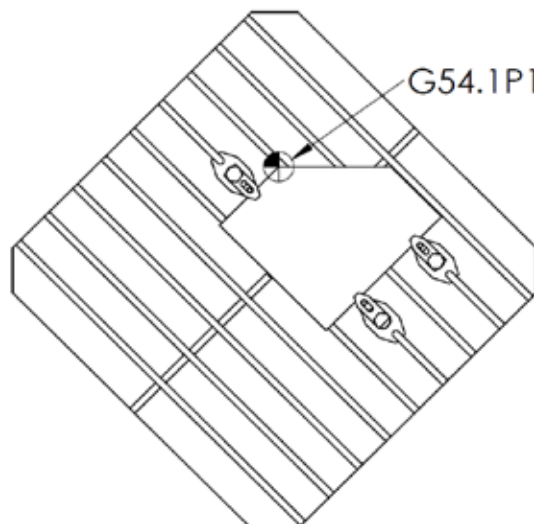
G54
G65P8001B45M3X225.5S1
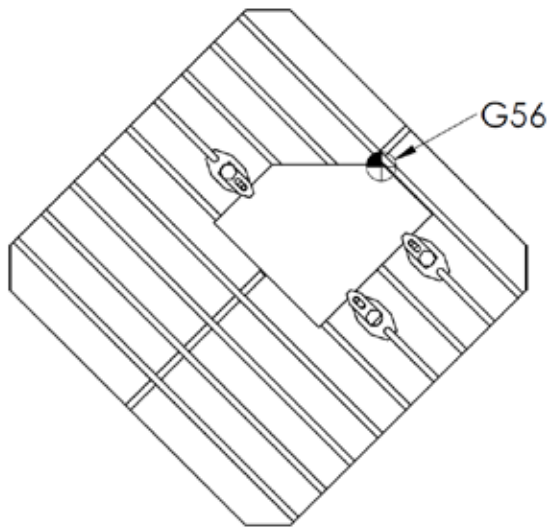
B45 TABLE

EXAMPLE N°3: creation of a new G54.1P1 offset obtained from a G54 offset but rotated by 45° and shifted by the values of X and Z necessary to bring the offset on the edge of the inclined plane. The X and Z values are translation values to be performed after the rotation, therefore setting the S parameter to zero.

G54
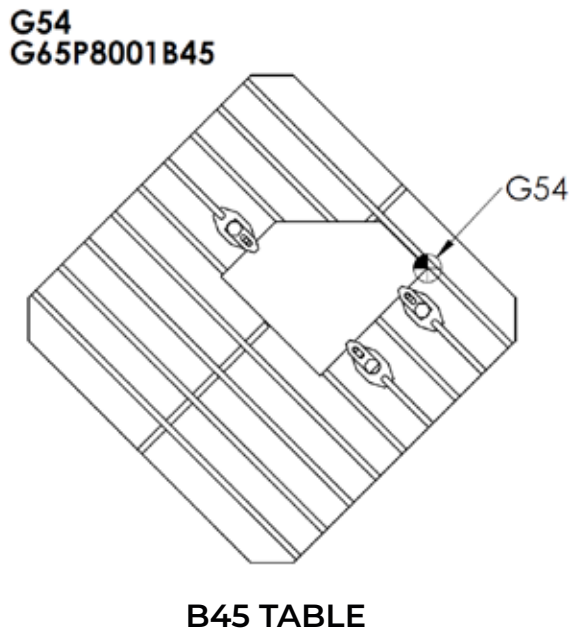**G65P8001B45M3X505.78Z-156.44S**

EXAMPLE N°4: Use this macro to rotate the G54 offset after rotating the table by 45°. Since no new offset is generated, the recalculation values are obtained by translating the G52 offset. Remember to deactivate the G52 offset at the end of the machining.

B45 TABLE

USE ADVICE

Which method to use between case 1 (creating a new offset) and case 2 depends very much on the machining to be carried out and the machine itself. Generally speaking, we can say that using case 1 (CREATION OF A DIFFERENT OFFSET) makes it possible to have two distinct offsets, with the benefit of being able to act on one or the other independently. For example, this frequently occurs on boring machines where in finishing there is a need to align between faces with very tight geometric tolerances. Small inaccuracies linked to the centre of the table or thermal deformations are then corrected on the individual offsets.

In the case of small or medium-sized horizontal machining centres, case 2 is much more common, where the same offset is always maintained, but simply shifted at each table rotation.

Usually, if case 1 is used, it is advisable to set the offset creation at the start of the program and possibly set a skip condition to be disabled only when the recalculation is required.

Let's look at a program example:

```
%
O0002

#100=0 (SET THE #100 PARAMETER TO 1 TO SKIP THE OFFSET
RECALCULATION)
IF[#100EQ1]GOTO9999

(****OFFSETS RECALCULATION FOR OTHER TABLE POSITIONS****)
G54
G65P8001B240M2
G65P8001B180M101
(****END OFFSETS RECALCULATION****)
N9999

(----START PROGRAM----)

G90G17G40

(****T2 - MILL D.100****)
T2 M6

G54G0B0
S640F5000M3
G0X0Y0
G0G43H1Z800
(B0 MACHINING)
G0 Z800

(---B240 TABLE---)
G55G0B240
G0X0Y0Z300
(MACHINING)
G0Z800

(---B180 TABLE---)
G54.1P1G0B180
G0X0Y0Z300
(MACHINING)
G0Z800

(---B15 TABLE---)
G54
G65P8001B15
G0B15
G0X0Y0Z300
(MACHINING)
```

OFFSET CREATION

FIRST MACHINING
WITH B0 OFFSET

B240 TABLE,
OFFSET ALREADY
RECALCULATED AT THE
START OF THE PROGRAM

B180 TABLE,
OFFSET ALREADY
RECALCULATED AT THE
START OF THE PROGRAM

B15 TABLE USING THE
SAME OFFSET MADE AT
B0, THEREFORE G54

```
G0Z800
G52X0Y0Z0

(--B35 MACHINING--)
G65P8001B-35X-300Y-100Z150S1
G0B-35
G0X0Y0Z300
(MACHINING)
G0Z800
G52X0Y0Z0
M30
%
```

B35 TABLE USING
THE SAME OFFSET
MADE AT B0,
THEREFORE G54
BUT NOT ONLY
ROTATED BUT ALSO
TRANSLATED

! WARNINGS

1. The subprogram contains two parameters for defining the position of the table centre. The first time the macro is loaded into the machine, it is necessary to modify the assignment of variables #127, #128 and #129 with their table centre values. The values of parameters #127, #128 and #129 are inside the macro itself in the first few lines. Without these correct values, the macro cannot correctly calculate the offsets.
EXAMPLE:
(****GENERAL MACHINE PARAMETERS****)
#127=-0.1(CENTRE POS. IN MCS TABLE IN X)
#128=0 (CENTRE POS. IN MCS TABLE IN Y)
#129=-2334.2(CENTRE POS. IN MCS TABLE IN Z)

2. The macro is delivered configured for a machine where the Z axis is horizontal, so the coordinates to be rotated are X and Z. If your machine has a vertical Z-axis, then simply set the internal parameter #131=1 and the macro will count the new offset with X and Y.

3. The macro is delivered with table degrees that correspond to the ISO standards. However, it may happen that the table rotation does not correspond to these. In this case, set parameter #130 equal to -1.

4. It is recommended that when using the macro for the first time, to always check the recalculated offsets to verify the accuracy of the table centre positions and the calculations of the macro.

5. The macro cannot be used with the modal activation function G66 but only with the G65 function, as explained in the manual.

6. To be able to use the macro on your machine, make sure that the parametric programming option is enabled. Although most machines have B macro programming enabled, check that your machine does too.

To do this, simply go to the MDI window and enter #100=1, press start and if no alarm message is emitted it means that programming with macro B is enabled. On some lathes in the 0 series, the # key may be missing, so to check this, simply load a program with only the line #100=1 and have it executed automatically. In this case too, no alarm message should appear.

Made in Italy

www.cncofcourse.com