

M-32 M-PLUS EIA/ISO PROGRAM USING MAZATROL TOOL DATA PAGE PARAMETER CHANGES

1.) F84 BIT #0 SET TO 1

This causes the tool length to be applied to the position display.

NOTE This has no effect on program operation and does not have to be changed if you do not wish the tool length to be applied to the position display.

2.) F92 BIT #7 SET TO 1

This will make the ACTUAL DIAMETER from the Mazatrol Tool Data page become valid for EIA/ISO programs. This diameter is used with the G41 or G42 cutter compensation commands. The control understands that this is a diameter value and will apply half of the diameter, or the radius, as the offset amount. Do not use a "D" value in the program.

Example: G42 G01 X3.0 Y0. F.006

3.) F93 BIT #3 SET TO 1

This makes the tool length from the Mazatrol Tool Data page valid for EIA/ISO programs. This tool length still requires the use of a G43 but do not use an "H" offset.

Example: G00 G43 Z1.0

4.) F94 BIT #2 SET TO 1

This stops the tool length from being canceled on a G28 or G30 command.

CAUTION If this parameter is not set to "1", and a G28 or G30 command is executed then the machine will try to move the spindle face to the current location of the tool tip.

M-PLUS CONTROLS ONLY

F94 BIT #7 SET TO 1

Makes the Mazatrol Tool Data Page Valid

Example Program

```
N10 G90 G80 G40 G00 G49;  
N20 T01T02 M06;  
N30 G54 X4.0Y-2.0;  
N40 S500 M03;  
N50 G43 Z1.0 M08;  
N60 Z-1.0;  
N70 G42 G01 X4.0 Y5.0 F.01;  
N80 X-4.0;  
N90 Y0;  
N100 X6.0;
```

```
N110 G40 G00 Z1.0 M09;  
N120 G91 G28 Z0;  
N130 G28 X0 Y0;  
N140 M01;  
(TO2 NEXT MACHINING OP)
```


M-32 M-PLUS EIA/ISO PROGRAM USING TOOL OFFSET PAGE PARAMETER CHANGES

1.) F84 BIT #0 SET TO 1

This causes the tool length to be applied to the position display.

NOTE This has no effect on program operation and does not have to be changed if you do not wish the tool length to be applied to the position display.

2.) F92 BIT #7 SET TO 0

This will let you use a "D" value in the program to designate an offset number from the Tool Offset page. This is to be used with a G41 or G42 in cutter compensation mode. The offset page uses 1/2 the cutter diameter or the radius value. Unless the program was generated centerline, then the offset value will start with a value of zero and you can offset +/- from there.

Example: G42 G01 X3.0 Y0. D31 F.006

3.) F93 BIT #3 SET TO 0

This makes the tool length from Tool Offset page valid for EIA/ISO programs. This tool length still requires the use of a G43 and also the use an "H" offset.

Example: G00 G43 Z1.0 H01

4.) F94 BIT #2 SET TO 1

This stops the tool length from being canceled on a G28 or G30 command.

CAUTION If this parameter is not set to "1", and a G28 or G30 command is executed then the machine will try to move the spindle face to the current location of the tool tip.

M-PLUS CONTROLS ONLY

F94 BIT #7 SET TO 0

Makes the Tool Offset Page Valid

EXAMPLE PROGRAM

```
N10 G90 G80 G40 G00 G40;  
N20 T01 T02 M06;  
N30 G54 X4.0 Y-2.0;  
N40 S500 M03;  
N50 G43 Z1.0 H01 M08;  
N60 Z-1.0;  
N70 G42 X4.0 Y5.0 D31 F.01;  
N80 X-4.0;  
N90 Y0;  
N100 X6.0;
```

```
N110 G40 G00 Z1.0 M09;  
N120 G91 G28 Z0;  
N130 G28 X0 Y0;  
N140 M01;  
(T02 NEXT MACHINING OP)
```