

IXION Deep-Drilling

M520

M530

Definition of Abbreviations and Variables

Description of cycle input with sequence

Description of cycles

Sample Programmes

Adr. (Address)

The address used for input of values in NC block

Vorz. (Prefix)

Each value should receive a prefix.

However, please note that positional addresses (R,Z,A,C,V,W) are absolute positions.

Incremental values (G91) are not operational in the programmes.

Form. (Form)

The form of the input e.g. 4.3 means 4 units in front of the comma and three units behind the comma.

(these are maximum values ! - smaller values are permitted)

Vari. (Variables)

The control has a memory for common (global) and local variables.

The local variables are operational only in the parts programme actually paging and in the next sub-programme level.

The variable numbers are attached to the addresses as per following table:

Addres	Local	Addres	Local
s	Variable	s	Variable
A	#1	S	#19
B	#2	T	#20
C	#3	U	#21
D	#7	V	#22
E	#8	W	#23
F	#9	X	#24
G	#10	Y	#25
H	#11	Z	#26
I	#4	-	#27
J	#5	-	#28
K	#6	-	#29
L	#12	-	#30
M	#13	-	#31
N	#14	-	#32
O	#15		
P	#16		
Q	#17		
R	#18		

A total of 4 sub-routine levels are permitted i.e.
the local variables #1-#32 4x are available in the controls.

The common (global) variables #100-#199 und #500-#599
can be read or described in all programme levels.
When commonly in operation, they have no divergent meanings.

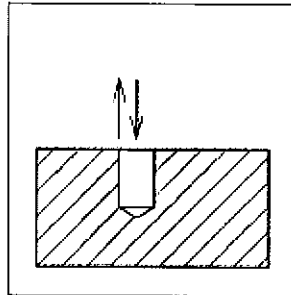
Double meaning is not permitted

The variables #100 - #199 are available for the machine
user.

**The variables #500 - #599 have fixed meanings given by
IXION and may not be used in the parts programmes.**

The following logic is maintained in the IXION Deep-Drilling cycles:
e.g. address W is attached to the local variable #23.
The local variable #23 is attached to the global variables #523, #553 und #583.
Jump size 30 was chosen to avoid overlaps; every variable number can thereby
be attached to its NC-programme address when checking the programme.

Twist drill **G181**

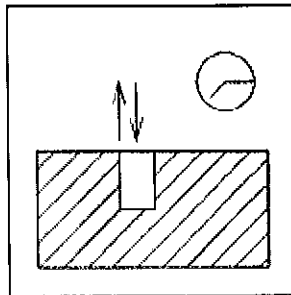


Adr.	Vorz.	Form	Vari.	Text	Remarks
R	+	4.3	18	Reference level	Distance from component
Z	-	4.3	26	Drilling depth	Final drilling depth
F	+	5	9	Feed	in mm/min
W	+	4.3	23	Return level	Level after drilling
S	+	5	19	Spindle speed	in min ⁻¹

Example:**G181 R2 Z-20 F40 W10 S400****Influence of programme monitoring : NONE****Sub-programmes used : O9281****Sequence:**

1. Travel in G90 (absolute)
2. Z-axis goes into position in reference level in G0
3. Z-axis goes into position in drilling depth in G1
(Drilling feed in mm/min)
4. Z-axis goes into position in reference level in G0
5. If G99 programmed: Jump to 7.
6. Z-axis goes into position in return level
7. Go back to position in G90 or G91

Boring / Reaming G182



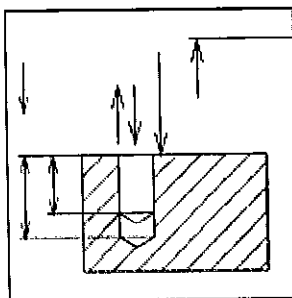
Adr.	Vorz.	Form	Vari.	Text	Remarks
R	+	4.3	18	Reference level	Distance from component
Z	-	4.3	26	Drilling depth	Final drilling depth
F	+	5	9	Feed	in mm/min
E	+	5	8	Return feed	in mm/min ; if 0 then in rapid feed
D	+	2.1	7	Dwelling time	in sec. ; Feed dwells in bottom of bore-hole
W	+	4.3	23	Return level	Level after drilling
S	+	5	19	Spindle speed	in min ⁻¹

Example:
G182 R2 Z-20 F40 E80 D0 W10 S400
Influence of process monitoring : NONE
Sub-programmes used: O9282

Sequence:

1. Travel in G90 (absolute)
2. Z-axis goes into position in reference level in G0
3. Z-axis goes into position in drilling depth in G1
(F = drilling feed in mm/min)
4. Switch to G0
5. If E0 programmed: Jump to 7.
6. Activate G1 and feed value if E unequal to 0
7. Lapse of dwelling time
8. Z-axis goes into position in reference level
9. Switch to G0
10. If G99 programmed: Jump to 12.
11. Z-axis goes into position in return level
12. Go back to position in G90 or G91

Deep-Drilling with Twist Drill G183



Adr.	Vorz.	Form	Vari.	Text	Remarks
R	+	4.3	18	Reference level	Distance from component
Z	-	4.3	26	Drilling depth	Final drilling depth
F	+	5	9	Feed	in mm/min
A	-	4.3	1	minimum advance	smallest drill advance; if $U < A$ then $A = U$
B	+	0.2	2	Reduction factor	Value to be multiplied with U ; if $B > 1$ then set $B = 0.9$
U	-	4.3	21	1st drilling depth	
C	+	1.3	3	Safety distance	Distance from material when re-entering the bore-hole
M	+	1	13	Coolant M7/M8	7: Inner-, 8: Outer-coolant ; if not 7 then set to 8
T	+	4	20	Replacement tool	if 0 then same tool used
W	+	4.3	23	Return level	Level after drilling
S	+	5	19	Spindle speed I	in min^{-1}
Q	+	5	17	Process number	1-99, not only the selected but also the following process number is used

Example:

G183 R2 Z-50 F30 A-3 B0.8 U-15 C0.5 M8 T1 W5 S400 Q1

Influence of process monitoring : Additional swarf clearance

Sub-programmes used : O9283,O9383

Sequence:

1. Travel in G90 (absolute)
2. Z-axis goes into position in reference level in G0
3. Z-axis goes into position in 1st. drilling depth in G1
(F = drilling feed in mm/min)
4. Swarf clearance in reference level in G0
5. Approach 1st drilling depth while considering safety distance
6. Calculate 2nd. drilling depth (multiplication of 1st)
drilling depth with reduction factor)
7. Approach 2nd drilling depth in G1
8. Continue calculating and positioning until final
drilling depth is reached
9. Switch to G0
10. If G99 programmed: Jump to 12.
11. Z-axis goes into position in return level
12. Go back to position in G90 or G91

The additional swarf clearances are counted and if the sum is greater than eight then the replacement of the tool is requested

The number of swarf clearances with respect to torque apexing is defined in variable #595 in programme O9383.

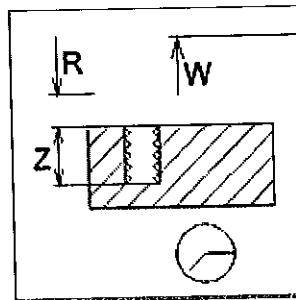
If no replacement tool (T0) is defined, then the length of the old tool is used.

Please note that length corrections for replacement tools have to be entered prior to machining.

If a replacement tool (T unequal to 0) is prescribed then length correction is automatically activated on continuation of programme.

In this case the depth of the pilot hole is taken into account.

Thread-cutting G184



Adr.	Vorz.	Form	Vari.	Text	Remarks
R	+	4.3	18	Reference level	Distance from component
Z	-	4.3	26	Drilling depth	Final drilling depth
F	+	5	9	Feed	in mm/rotation.; = pitch
M	+	1	13	Rotational direction	if 4 then left-hand thread otherwise right-hand thread
D	+	2.1	7	Dwelling time	must be 0 otherwise the feed dwells on base of bore-hole and the spindle continues rotating
W	+	4.3	23	Return level	Level after drilling
S	+	5	19	Spindle speed	in min ⁻¹

Example:

G184 R5 Z-25 F1.5 M3 D0 W10 S400

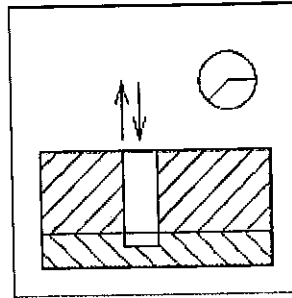
Influence of process monitoring : NONE

Sub-programmes used : O9284

Sequence:

1. Travel in G90 (absolute)
2. Selection of right- or left-hand thread via choice in rotational direction of spindle (M)
3. Z-axis goes into position in reference level in G0
4. Z-axis goes into position in drilling depth (drilling feed in mm/rotation = pitch)
5. Z-axis goes into position in reference level (drilling feed in mm/rotation = pitch)
6. If G99 programmed: Jump to 8.
7. Z-axis goes into position in return level
8. Go back to position in G90 or G91

Gundrilling G185



Adr.	Vorz.	Form	Vari.	Text	Remarks
R	+	4.3	18	Reference level	Distance from component
Z	-	4.3	26	Drilling depth	Final drilling depth
F	+	5	9	Feed	in mm/min
S	+	5	19	Rpm 1	in min ⁻¹
U	-/+	4.3	21	Switch level 1	Switch-on point for spindle
E	+	5	8	Feed 2	if 0 then Feed 2 = Feed 1 and
					Rpm 2 is set = Rpm 1
H	+	5	11	Rpm 2	if 0 then Feed 2 = Feed 1 and
					Rpm 2 is set = Rpm 1
V	-	4.3	22	Switch level 2	if V > U then set V = U -1
T	+	4	20	Replacement tool	if 0 then same tool is used
W	+	4.3	23	Return level	Level after drilling
Q	+	5	17	Process number	1-99, not only the selected but also the following process number is used.
					On account of differing Feed/Rpm

Example:

G185 R2 Z-500 F30 S2000 U-25 E70 H2500 V-35 T5 W10 Q6

Influence of process monitoring: Dwelling time

Sub-programmes used: O9285.O9385

WARNING !

Under no circumstances may you start-up a tool without there first being a guide.

The high-pressure coolant pump is only started under this cycle.

It is not permitted to programme M-function M11 outside the cycle.

When working with press-fit drill bushings/press fit drill arm the press-fit drill bushing is switched on by ADJUSTMENT in the LIGHT mode and by TOOL/MENU/PLC-SW in the STANDARD (conventional programming) mode.

When working without press-fit drill bushing it must be switched off.

Sequence:

1. Travel in G90 (absolute)
2. Z-axis goes into position in reference level in G0
3. Z-axis goes into position in 1st switch over level at feed 3000 mm/min
4. if available : press-fit drill bushing is pressed up against the component
5. Elapse of 2 seconds dwelling time
6. High-pressure coolant pump is activated
7. Spindle starts up at the rpm programmed under S
if Feed 2 or Rpm2 = 0
Z-axis goes into position in drilling depth with feed F jump to 12
8. Z-axis goes into position in 2nd switch level with feed F
9. programmed process number is increased by 1
10. Spindle is switched to Rpm as programmed under H
11. Z-axis goes into position in drilling depth with feed E
12. Switch to G0, spindle is switched off
13. Coolant is switched off
14. If G99 programmed: Jump to 16.
15. Z-axis goes into position in return level
16. Withdrawal of press-fit drill bushing
17. Elapse of 2 seconds dwelling time
18. Go back to position in G90 or G91

If one of the limits in the torque or, optionally , feed force monitor are superceded then the feed movement is interrupted and a dwelling time elapses.

This dwelling time is used for washing the swarf out of the bore-hole.

The dwelling time and the number of dwelling times are defined in programme O9385.

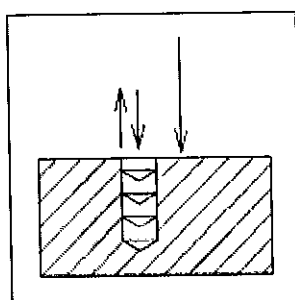
Dwelling time: in NC-block G4 X*
 (*=dwelling time in seconds)

Number of dwelling times: in NC-block IF[#595LT*]GOTO
 (*=number of dwelling times)

Programming of 2 different feeds and spindle speeds has been made available for following purposes:

1. Machining of different materials in one drilling operation.
2. Slow initial drilling on entry of tool into the material.

Twist drill with swarf breakage (Woodpecking) G186



Adr.	Vorz.	Form	Vari.	Text	Remarks
R	+	4.3	18	Reference level	Distance from component
Z	-	4.3	26	Drilling depth	Final drilling depth
F	+	5	9	Feed	in mm/min
U	-	4.3	21	1st drilling depth	
C	+	1.3	3	Return value	Withdrawal from base of bore-hole
M	+	1	13	Coolant M7/M8	7: Inner-, 8: Outer coolant; if not 7 then set to 8
T	+	4	20	Replacement tool	if 0 then same tool us used
W	+	4.3	23	Return level	Lift off after drilling
S	+	5	19	Spindle speed I	in min ⁻¹
Q	+	5	17	Process number	1-99, not only the selected but also the following process number is used

Example:

G186 R2 Z-50 F30 U-10 C1 M8 T1 W5 S400 Q5

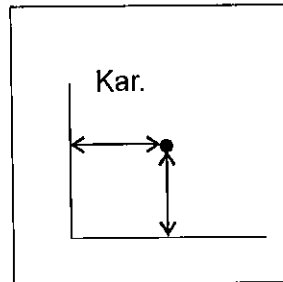
Influence of process monitoring : Additional swarf clearance

Sub-programmes used: O9286,O9386

Sequence:

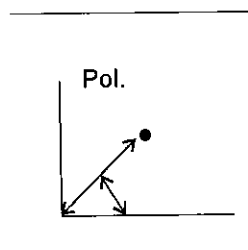
1. Travel in G90 (absolute)
2. Z-axis goes into position in reference level in G0
3. Z-axis goes into position in 1st drilling depth in G1
(F = drilling feed mm/min)
4. Retract Z-axis by return value
5. Approach final drilling depth
6. Continue calculating and positioning until final drilling depth is reached
7. Switch to G0
8. If G99 programmed: Jump to 10
9. Z-axis goes into position in retract level
10. Go back to position in G90 or G91

Single hole cart.



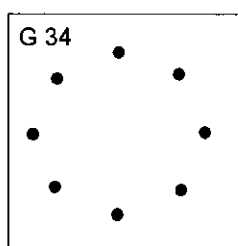
Adr.		Form	Vari.	Text
X	+ -	4.3	24	X-Coordinate
Y	+ -	4.3	25	Y-Coordinate

Single hole polar



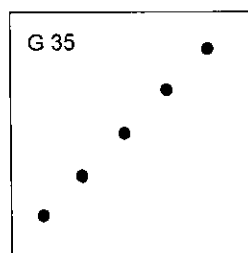
Adr.		Form	Vari.	Text
I	+ -	4.3	4	radius
J	+ -	4.3	5	angle

Full circle of holes G34



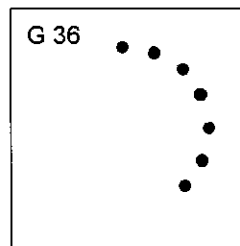
Adr.		Form	Vari.	Text
X	+ -	4.3	24	X-Centre point
Y	+ -	4.3	25	Y-Centre point
I	+	4.3	4	radius
J	+ -	4.3	5	Angle 1. Hole
K		4	6	No.of holes

Row of holes G 35



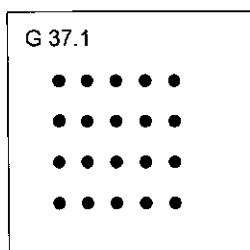
Adr.		Form	Vari.	Text
X	+ -	4.3	24	X-Coordinate 1. hole
Y	+ -	4.3	25	Y-Coordinate 1. hole
I	+ -	4.3	4	Distance
J	+ -	4.3	5	Angle
K		4	6	No. of holes

Circle of holes G36



Adr.		Form	Vari.	Text
X	+ -	4.3	24	X-centre point
Y	+ -	4.3	25	Y-centre
I	+	4.3	4	Radius
J	+ -	4.3	5	Angle 1. hole
P	+ -	4.3	16	Angle between holes
K		4	6	No. of holes

Grid of holes G37.1



Adr.		Form	Vari.	Text
X	+ -	4.3	24	X-coordinate 1. hole
Y	+ -	4.3	25	Y-coordinate 1. hole
I	+	4.3	4	Distance X-Axis
P		4	16	No. of holes X
J	+ -	4.3	5	Distance Y-Axis
K		4	6	No. of holes Y

Variables applied for IXION Cycles

#500	#525	#550	#575	#1033
#501	#526	#551	#576	#1034 Process number
#502	#527	#552	#577	#1100 Spindle Limit 1
#503	#528	#553	#578	#1101 Spindle Limit 2
#504	#529	#554	#579	#1102 Z Axis Limit 1
#505	#530	#555	#580	#1103 Z Axis Limit 2
#506	#531	#556	#581	#1104 Upper Oil Pressure Level
#507	#532	#557	#582	#1105 Lower Oil Pressure Level
#508	#533	#558	#583	#1106 Upper Oil Flow Level
#509	#534	#559	#584	#1107 Lower Oil Flow Level
#510	#535	#560	#585	#1116 Press-fit bush active
#511	#536	#561	#586	#1124 TL-Light Mode active
#512	#537	#562	#587	#1125 Set text of re- placement tool
#513	#538	#563	#588	#1126 Single Axis CNC (M3)
#514	#539	#564	#589	#1127 Toolchanger Supplied
#515	#540	#565	#590	#1134 Tool counter for auto-toolchanger
#516	#541	#566	#591	#4003 G90/G91
#517x	#542	#567	#592	#4010 G98/G99
#518x	#543	#568	#593	#5001 Position Single axis
#519x	#544	#569	#594	#5003 Position 3 axes
#520x	#545	#570	#595x	
#521x	#546	#571	#596x	
#522x	#547	#572	#597	
#523x	#548x	#573x	#598	
#524	#549	#574	#599x	

General Commandos in Programmes:

M0	Halt programme until NC-start
M1	Selected halt
M2	End of programme
M30	End of programme

Command of Spindle:

M3	Clockwise Spindle
M4	Anticlockwise Spindle
M5	Stop Spindle
M19	Orientate spindle

Toolchange:

*M6	Automatic toolchange
*M60	Manual toolchange

Coolant:

M7	Internal coolant on
M8	External coolant on
M9	Coolant off
*M10D%	Pressure level to NC programme
M11	High pressure coolant on

Drill Bush Carrier:

M31	Press-fit bushing to workpiece
M32	Retract press-fit bushing from workpiece

Quill:

M33	Clamp quill
M34	Unclamp quill

Jumps:

M98Pxxxx	Jump to subroutine xxxx
M99Pxxxx	Jump back to main programme, to block no.xxxx
M96P	Defined jump (if torque is responding)
M97	Cancel defined jump

Processs Monitoring:

M21	Activate interruption on response from monitoring signal
M22	Stop interruption on response from monitoring signal
M101 up to M199	Activate monitoring using process number Process number= M-function number -100 e.g. M102 means select the process number 2
M200	End of monitoring

* defined by M-macros

Solution of problems

When the message „**tool-for-2.time**“ appears, you have the following possibilities :

-If you again want to use the tool which is in the spindle press to reset the program.



Or due to the torque monitor it comes for the second time to the tool there are 3 possibilities:

1. press  to stop the program.
2. slightly regrind the drill and press  to reset the program.
3. insert a newly measured drill and book the tool length in the length correction chart(table) under the number corresponding to the replacement tool. Then press  to reset the program.

1-axis machines

ATTENTION

When switching from mode **LIGHT** ()
to mode **STANDARD** (, , , , ) ,
G54 must close cycle

O 101 (Pilot drilling)
M60 T1
G0 G43 H1 Z5
G181 R2 Z-2 F60 W5 S900
M30

Programme number; with commentary
Tool change with tool number
Rapid feed, length corr., tool no., position
Twist drill cycle
Programme end

O 102 (Ø 9.8 / with chip breaking)
M60 T2
G0 G43 H2 Z5
G186 R2 Z-30 F40 U-6 C0.5 M8 T0 W5
S600 Q1
M30

Programme number; with commentary
Tool change with tool number .
Rapid feed, length corr., tool no., position
Twist drill with woodpecking cycle

Programme end

O 103 (Ø 10mm / Reiben)
M60 T3
G0 G43 H3 Z5
G182 R2 Z-27 F40 E100 D0 W5 S300
M30

Programme number; with commentary
Tool change with tool number
Rapid feed , length corr., tool no., position
Reaming cycle
Programme end

O 104 (Ø 10 / 400 / Gundrill)
M60 T4
G0 G43 H4 Z5
G185 R2 Z-400 F40 S2000 U-25 E80
H2200 V-33 T0 W5 Q3
M30

Programme number; with commentary
Tool change with tool number
Rapid feed, length corr., tool no., position
Gundrilling cycle

Programme end

O 105 (M12 / Thread)
M60 T5
G0 G43 H5 Z10
G184 R5 Z-25 F1.75 M3 D0 W10 S250
M30

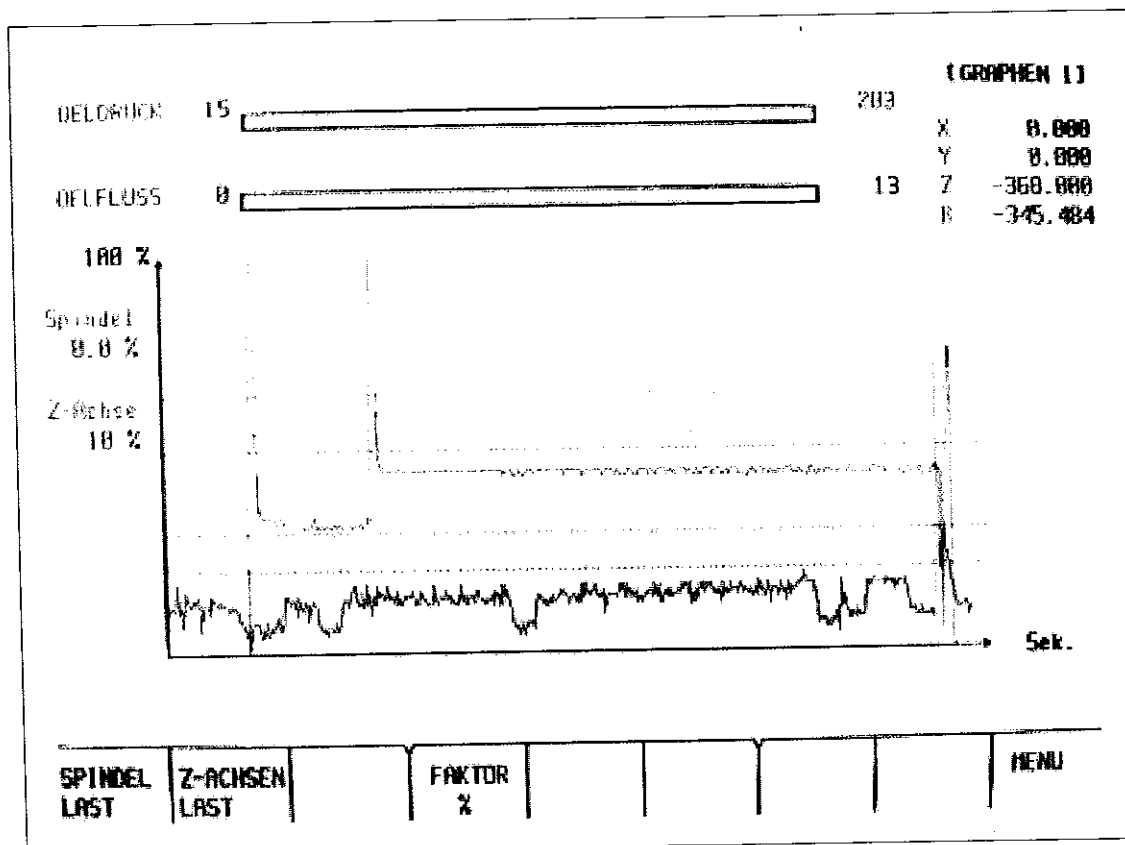
Programme number; with commentary
Tool change with tool number
Rapid feed, length corr., tool no., position
Thread-cutting cycle
Programme end

For 4-Axis machines

O 201 (Thread G1/4" and M20)	Programme No. and commentary
M60 T10	Tool change with tool number. NC-Pilot Drill Ø 10 mm
G56	Activate zero point shift
G0 W0	Rapid feed with W-travel to zero
G43 H10 Z5	Tool offset with tool position
G181 R2 Z-2 F60 W5 S900	Twist drill cycle
M98 P202	Call up subroutine (drilling patterns)
M98 P203	Call up subroutine (drilling patterns)
G67	Cancel drilling cycle
M60 T11	Tool change with tool number. Drill Ø 11,5 mm
G56	Activate datum shift
G0W0	Rapid feed with W-travel to zero
G43 H11 Z5	Tool offset with tool position
G183 R2 Z-35 F40 A-4 B.8 U-10 C.5 M8	Deep-drilling cycle for twist drills
T12 W5 S600 Q11	
M98 P202	Call up subroutine (drilling pattern)
G67	Cancel drilling cycle
M60 T13	Tool change with tool number Drill Ø 17,5 mm
G56	Activate datum shift
G0W0	Rapid feed with W to zero
G43 H13 Z5	Tool length offset with tool position
G183 R2 Z-40 F40 A-4 B.8 U-10 C.5 M8	Deep-drilling cycle for twist drills
T14 W5 S600 Q11	
M98 P203	Call up subroutine (drilling patterns)
G67	Cancel drilling cycle
M60 T15	Tool change with tool number Thread G 1/4
G56	Activate datum shift
G0W0	Rapid feed with W axis to zero
G43 H15 Z10	Tool length offset with tool position
G184 R5 Z-25 F1.337 M3 D0 W10 S400	Thread cutting cycle
M98 P202	Drilling cycle (drilling pattern)
G67	Cancel drilling pattern
M60 T16	Tool change with tool number. Thread M20

G56	Activate datum shift
G0W0	Rapid feed with W axis to zero
G43 H16 Z10	Tool length offset with tool position
G184 R5 Z-30 F2.5 M3 D0 W10 S300	Thread cutting cycle
M98 P203	Call up subroutine (drilling pattern)
G67	Cancel drilling cycle
M30	End of programme
O 202 (Bohrbild fuer G1/4)	Programme number with commentary
G35 X177 Y68 I20 J0 K1	Row of holes
X30 Y30	Individual bore
M99	End of subroutine
O 203 (Bohrbild fuer M20)	Programme number with commentary
X25 Y45	Individual bore
M99	End of subroutine

Process Control for Mitsubishi Monitor M 520



0511-001

IXION GmbH & Co. KG
Postfach 70 08 29
22008 Hamburg

Jenfelder Str. 30
22045 Hamburg

Service

Tel. 040 / 66 98 09 - 76

Fax 040 / 66 98 09 - 32

Start-up Settings

Display Menu	3
Start-up Settings	3
Input values	3

Process Table

Process Table M325	6
Process Table M3	6

Graphs

Graphs Overview (Example M3)	7
------------------------------------	---

Adjustments

Adjustments - Overview	9
Menu Keys	10
SPS-Monitor	10

Start-up Settings

Menu-M325

ACTUAL PROCESS NUMBER :			
#1	Z-AXIS	OBSERVATION	: #11 DELAY TIME LEARNING : 123 SEK
#2	SPINDLE 1	OBSERVATION	: #12 DELAY TIME OIL PRESSURE : 123 SEK
#3	SPINDLE 2	OBSERVATION	: #13 DELAY TIME OEL PRESSURE : 123 SEK
#4	OIL PRESS 1	OBSERVATION	: #14 MINIMUM TIME LEARNING : 123 SEK
#5	OIL PRESS 2	OBSERVATION	: #15 Z-LEVEL 1 DEFAULT : 123 %
#6	OIL FLOW 1	OBSERVATION	: #16 Z-LEVEL 2 DEFAULT : 123 %
#7	OIL FLOW 2	OBSERVATION	: #17 S1-LEVEL 1 DEFAULT : 123 %
#8	TOLERANCE	CANCEL	: #18 S1-LEVEL 2 DEFAULT : 123 %
#9			: #19 OIL PRESS 1 UPPER LEVEL : 123 %
#10			: #20 OIL PRESS 1 LOWER LEVEL : 123 %
			: #21 OIL FLOW 1 UPPER LEVEL : 123 %
			: #22 OIL FLOW 1 LOWER LEVEL : 123 %
			: #23 S2-LEVEL 1 DEFAULT : 123 %
			: #24 S2-LEVEL 2 DEFAULT : 123 %
			: #25 OIL PRESS 2 UPPER LEVEL : 123 %
			: #26 OIL PRESS 2 LOWER LEVEL : 123 %
			: #27 OIL FLOW 2 UPPER LEVEL : 123 %
			: #28 OIL FLOW 2 LOWER LEVEL : 123 %
ACTUAL OIL PRESSURE 1 : BAR			
ACTUAL OIL FLOW 1 : l/min			
ACTUAL OIL PRESSURE 2 : BAR			
ACTUAL OIL FLOW 2 : l/min			

setupm325d.xls

Menu-M3

ACTUAL PROCESS NUMBER :		: ACTUAL OIL PRESSURE: 100 BAR	
		: ACTUAL OIL FLOW: 123 l/min	
#1	SPINDLE CONTROL	: #11 DELAY TIME LEARNING	: 123 SEK
#2	Z-AXIS CONTROL	: #12 DELAY TIME OIL PRESSURE	: 123 SEK
#3	OIL PRESSURE CONTROL	: #13 DELAY TIME OEL PRESSURE	: 123 SEK
#4	OIL FLOW CONTROL	: #14 MINIMUM TIME LEARNING	: 123 SEK
#5	TOLERANCE CANCEL	: #15 S-LEVEL 1 DEFAULT	: 123 %
#6		: #16 S-LEVEL 2 DEFAULT	: 123 %
#7		: #17 Z-LEVEL 1 DEFAULT	: 123 %
#8		: #18 Z-LEVEL 2 DEFAULT	: 123 %
#9		: #19 OIL PRESS UPPER LEVEL	: 123 %
#10		: #20 OIL PRESS LOWER LEVEL	: 123 %
		: #21 OIL FLOW UPPER LEVEL	: 123 %
		: #22 OIL FLOW LOWER LEVEL	: 123 %
# ()	DATA ()		
		OIL PRESSURE	OIL FLOW
		MENU	

setupm3d.xls

Display Menu

- press "F0" key
- press "F2" key,
- press "Settings" key

The menu is displayed and several menu selections are highlighted in color. Only these monitoring options are active.

Start-up Settings

- **Example: activate the spindle monitoring and Z-axis monitoring functions of the Mitsubishi-M3 Monitor**
- using the arrow keys, place the cursor in the last line
() DATA (),
- press key 1 and **INPUT**
(2) is displayed,
- press **INPUT**,
Both Monitors are now active.

Input values

Starting from #11, values must be input for each individual item.

- Using the arrow keys, move the cursor to menu item # 11.
(11) DATA () is displayed in the last line.
- enter the value, press **INPUT**



We recommend inputting the values indicate after the word "settings"

- **# 11 LEARN TIME-LAG**

Setting: 3 sec.

This delay is necessary to prevent learning of peak loads at the beginning of the cutting cycle.

- **# 12 OIL PRESSURE TIME-LAG**

Setting: 2 sec.

- **# 13 OIL FLOW TIME-LAG**

Setting: 2 sec.

If the drill encounters cross drills while milling, greater deviations in oil flow and oil pressure readings may arise. Here monitoring should not yet respond.

Deviations are ignored for 2 seconds.



In general, for #12 and #13, select the shortest possible times required.

- **# 14 LEARN MINIMUM TIME**

Setting: 5 sec.

Here, the value must exceed the value entered in #11.

If the "learning active" switch is only briefly activated, the monitor learns for the duration of the time indicated in #14.

- **# LEVEL 1**

...LEVEL 1 is the dull drill warning. This setting depends on the material being processed and the drill diameter.

Setting: between 140% and 170%

Should the spindle load exceed the set level, forward feeding continues for 2 seconds. The monitoring system re-tests whether spindle load again exceeds the set level. If so, the process is repeated 6 times per drilling depth. Thereafter, the drilling process is interrupted and the drill moves to the tool replacement position. Replacement tool can then be installed.

- **# LEVEL 2**

...LEVEL 2 is the overload limit for the drill.

Setting: a value should be input that is 20% to 40% greater than the value in LEVEL 1

Should the spindle load exceed the value set, the drilling process is immediately interrupted. The drill moves to the tool replacement position.

- Check the tool for tool damage.
if none,
- Install replacement tool and complete drill job.

- **# UPPER LIMIT**

Setting: between 110% and 140%

Should the UPPER LIMIT exceed the set value, the drill process is interrupted after a LAG-TIME of 2 seconds. The drill moves to tool replacement position.

- Check the tool for tool damage
if none,
- Install replacement tool and complete drill job.

LOWER LIMIT

Setting between 90% and 70%

Should the ...LOWER LIMIT fall short of the level set, the drilling process is interrupted after a LAG-TIME of 2 seconds. The drill moves to the tool replacement position

- Check the tool for tool damage
if none,
 - Install replacement tool and complete drill job.
-
- **Oil Flow**
With the **OIL FLOW** menu key, the current value displayed above for oil flow can be copied to the process table.
 - **Oil Pressure**
With the **OIL PRESSURE** menu key, the current value displayed above for oil pressure can be copied to the process table.

Process Table M325

PROCESS NUMBER :		PROCESS TABLE			
TOOL NUMBER :					
FEEDRATE : mm/min					
SPINDLESPEED : 1/min					
SPINDLE 1 : %	Z-AXIS : %	OIL PRESS 1 : BAR	OIL FLOW 1 : l/min		
LEVEL 1 : %	LEVEL 1 : %	LEVEL 1 : BAR	LEVEL 1 : l/min		
LEVEL 2 : %	LEVEL 2 : %	LEVEL 2 : BAR	LEVEL 2 : l/min		
SPINDLE 2 : %		OIL PRESS 2 : BAR	OIL FLOW 2 : l/min		
LEVEL 1 : %		LEVEL 1 : BAR	LEVEL 1 : l/min		
LEVEL 2 : %		LEVEL 2 : BAR	LEVEL 2 : l/min		

prozeßm325d.xls

Process Table M3

PROCESS NUMBER :		PROCESS TABLE			
TOOLNUMBER :		PROGRAM NUMBER :			
FEEDRATE : mm/min		DATE :			
SPINDLESPEED : 1/min		TIME :			
SPINDLE : %	Z-AXIS : %	OIL PRESS: BAR	OIL FLOW: l/min		
LEVEL 1 : %	LEVEL 1 : %	UPPER LEVEL: BAR	UPPER LEVEL: l/min		
LEVEL 2 : %	LEVEL 2 : %	LOWER LEVEL: BAR	LOWER LEVEL: l/min		
NEW PROCESS NUMBER ()					
DELETE PROCESS			MENU		

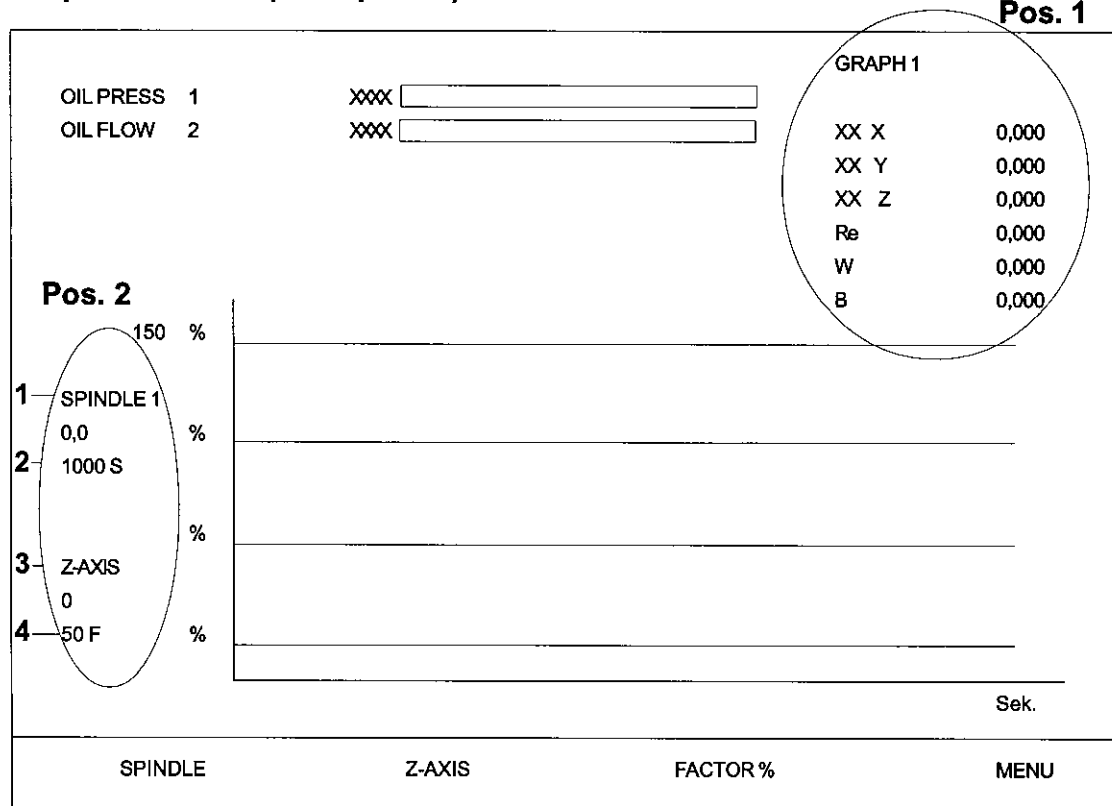
prozeßm3d.xls

The last process selected (M 1XX) / (Q XX) is displayed when this menu is called up. All the relevant data from a learned processing sequence are displayed in the process table and can be changed by the operator. Using the arrow keys, the bar cursor can be moved up or down. The respective input option is displayed in front of the input bracket. LEVEL 1 and LEVEL 2 can be entered in increments of 0.1. All other values are whole numbers.

• Menu Key

With the menu key **DELETE PROCESS**, the displayed process is deleted. All data are reset to "0".

Graphs Overview (Example M3)



graphend.xls

In the Graphs Menu, all monitorable data are graphically displayed in real time. Storing of data is not provided for.

- **Position 1 M3:**
The axis positions and the remaining distance on the Z axis are displayed.
- **Position 1 M325:**
Here, the axis positions are displayed from the Machine zero point.
- **Position 2 M3:**
 1. Half of the actual spindle motor performance is displayed graphically in percent.
 2. Spindle RPM "S" are displayed.
 3. The actual value of Z-axis motor performance is displayed.
 4. Present progress F is displayed in mm/min.
- **Position 2 M325:**
 1. Half of the actual spindle motor performance is displayed graphically in percent.
 3. The actual value of Z-axis motor performance is displayed.

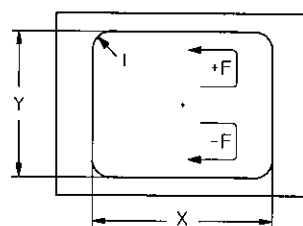
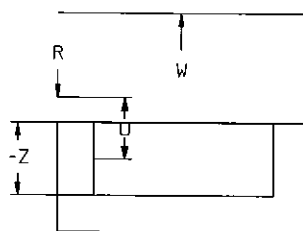
- **Oil Pressure, Oil Flow**

Oil pressure and oil flow are always displayed as a bar graph. On the left side of the graph, the current values for oil pressure and flow are displayed numerically. On the right, theoretical maximum values (at 10V analog input) are depicted as numerical data. The vertical lines above the bars indicated the upper and lower limits. (Only when the tolerance display is on)

- **Spindle, z-axis and Factor Menu Keys**

Spindle or z-axis loads can be turned off individually by means of Menu keys. They are displayed as line graphs. To the left of the graphs, current values for spindle and Z-Axis loads are displayed numerically. When tolerance display is turned on, Level 1 and Level 2 values are indicated as dotted lines. The graphs move from left to right across the diagram. When the end of the diagram is reached, the graphs are deleted and the display begins over. By means of the Factor % menu key, the maximum display can be changed in increments of 50% (150%, 100%, 50%).

Square Pocket G175



TLF1-100

Adr.	Sign.	Vari.	Text
R	+/-	18	reference level
Z	-	26	milling depth
F	+/-	9	milling feedrate
S	none	19	spindle speed
U	none	21	pecking depth
E	-/+	8	advance feed
X	none	24	length of the groove
Y	none	25	width of the groove
I	none	4	inner radius of the corner
W	+/-	23	retract level

Comment

distance upon workpiece
depth of the groove
+ (G3) CCW milling direction
- (G2) CW milling direction

Feedrate for pecking. With the negative sign the program run only the finishing operation.
value greater or equal then Y
value greater or equal then X
value greater then toolradius

z-level after milling operation

Used Subprogram : O9775

Finish overmeasure can modified in bloc N10 in the subprogram (standard 0.5mm). Starting point is in the centre of the left circle. It is possible to mill only horizontal pockets. For pockets under an angle is the coordinate rotating function of the Mitsubishi-control necessary. (G68/G69 Mitsubishi Option). If the E-parameter-value is negative the cycle runs automaticly the finishing operation only.

Example:

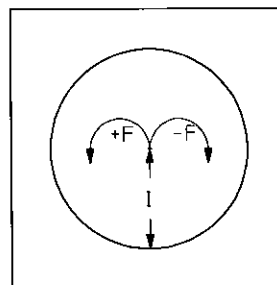
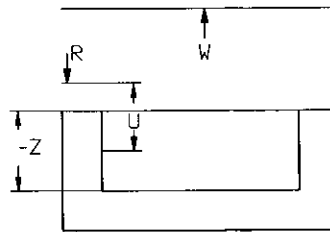
M6 T10
G55
G0 B0
G43H10D10Z5

G175 R2 Z-20 F300 S1500 U2 E150 X200
Y100 I10 W10
X100 Y20
X0 Y150
G67
M30

auto toolchange with tool number
zero shift active
rotary-table moves to zero-position (TLF)
tool length and radius compensation; Position of z-axis;
cycle definition

milling coordinates of the start point
milling coordinates of the start point
cycle call switch off
end of program

Round Pocked G176



TLF1-101

Adr.	Sign.	Vari.	Text
R	+/-	18	reference level
Z	-	26	milling depth
F	+/-	9	milling feedrate
S	none	19	spindle speed
U	none	21	peckind depth
E	-/+	8	advance feed
I	none	4	inner radius of the pocket
W	+/-	23	retract level

Comment

distance upon workpiece
depth of the groove
+ (G3) CCW milling direction
- (G2) CW milling direction

Feedrate for pecking. With the negative sign the program run only the finishing operation.
 $2 \times I$ = diameter of the pocket. Radius value greater then toolradius plus finish overmeasure.
Z-level after milling operation.

Used Subprogram: O9776

Finish overmeasure can modified in bloc N10 in the subprogram (standard 0.5mm). Starting point is in the centre of the pocked. It is posible to mill only horizontal pockets. For pockets under an angle is the coordinate rotating function of the Mitsubishi-control necessary. (G68/G69 Mitsubishi Option). If the E-parameter-value is negative the cycle runs automaticly the finishing operation only.

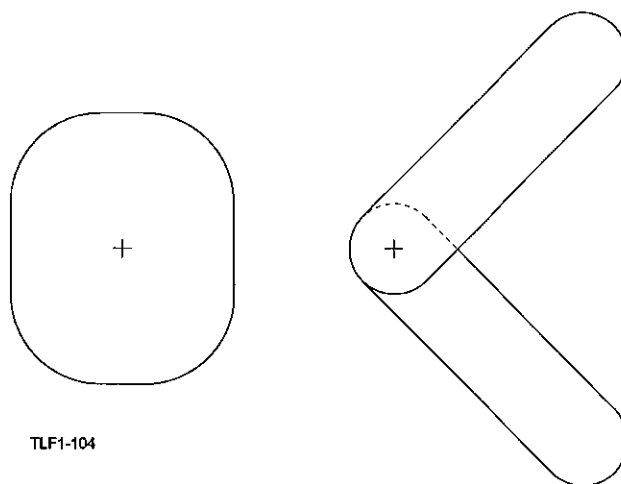
Example:

M6 T10
G55
G0 B0
G43H10D10Z5

G176 R2 Z-20 F300 S1500 U2 E150 I25 W10
X100 Y20
X0 Y150
G67
M30

auto toolchange with tool number
zero shift active
rotary-table moves to zero-position (TLF)
tool length and radius compensation; position of z-axis;
cycle definition
milling coordinates of the start point
milling coordinates of the start point
cycle call switch off
end of program

Partrotation Mitsubishi Care G68



Example:

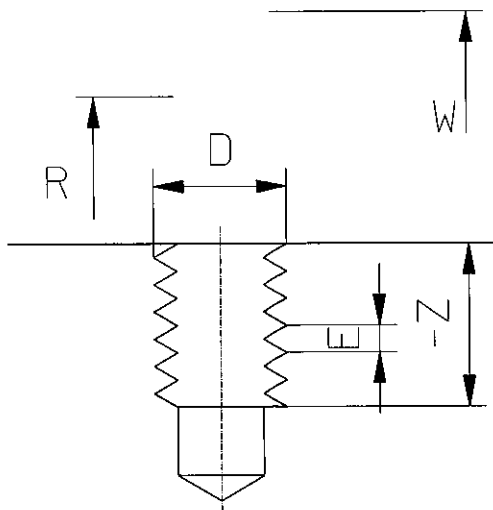
M6T10
 G55
 G0B0
 G43H10D10Z5
 G68X55Y70R90
 G175R2Z-20F-300S1500U7E150X60Y50I20W10
 X55Y70
 G67
 G69
 G68X133Y67R45
 G174R2Z-20F300S1500U7E150X80Y20W10
 X133Y67
 G68X133Y67R-45

 G67
 G69
 M30

automatic tool change with tool number
 activation zero shift
 table moves to zero
 tool compensation, length, radius, position
 partrotation with rotation centre and angle
 see above
 milling coordinates
 switch off milling cycle
 switch off partrotation
 partrotation with rotation centre and angle

 milling coordinates
 partrotation with rotation centre and angle
 (milling coordinates)
 switch off milling cycle
 switch off partrotation
 program end

Thread - Milling M284



TLF1-105

Adr.	Vorz.	Vari.	Text	Comment
R	+/-	18	reference level	distance upon workpiece
Z	-	26	depth of the thread	depth of the thread
F	+	9	milling feedrate	in mm/min
S	ohne	19	spindle speed	
E	+	8	pitch	
D	+	7	thread diameter	thread diameter
M	ohne	13	coolant	
W	+/-	23	retract level	Z-level after milling operation.

Used Subprogram: O9784

For this cycle is no IXION help function available.

Beispiel:

M6 T10
 G55
 G0 B0
 G43H10D10Z5
 R2 Z-20 F150 S1800 E4 D36 M8 W10
 X100 Y20
 X0 Y150
 G67
 M30

automatic tool change with tool number
 activation zero shift
 table moves to zero
 tool compensation, length, radius, positionM284
 see above
 milling coordinates
 milling coordinates
 switch off milling cycle
 program end

