MILLING MACROS

Hole patterns

Points (G1400)
Coordinates for the holes can be specified in the following way:

\[ G1400 \ X_\ Y_\ A_\ B_\ C_\ D_\ E_\ F\ldots \]

where

- Maximum 8 points can be specified.
- The following table shows the addresses of X, Y coordinates of the holes.

<table>
<thead>
<tr>
<th>Points</th>
<th>Coordinate X</th>
<th>Coordinate Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point 1</td>
<td>X</td>
<td>Y</td>
</tr>
<tr>
<td>Point 2</td>
<td>A</td>
<td>B</td>
</tr>
<tr>
<td>Point 3</td>
<td>C</td>
<td>D</td>
</tr>
<tr>
<td>Point 4</td>
<td>E</td>
<td>F</td>
</tr>
<tr>
<td>Point 5</td>
<td>H</td>
<td>I</td>
</tr>
<tr>
<td>Point 6</td>
<td>J</td>
<td>K</td>
</tr>
<tr>
<td>Point 7</td>
<td>R</td>
<td>S</td>
</tr>
<tr>
<td>Point 8</td>
<td>T</td>
<td>U</td>
</tr>
</tbody>
</table>

Sample:
%O2001(Points)
M3 S1000 G91 G53
Start spindle speed specified by address S.
Programming of incremental data
Positioning in machine coordinate system.

G83 Z-20 R-5 E0.5 P1 F100 L0
Setting drilling cycle data without execution of drilling.

G1400 X170 Y120 A180 B40 C50 D30
Setting 3 holes by X-Y coordinates.

G80
Canned cycle cancel.

M5
Stop spindle.

%
G1401 Hole pattern: Positions at same spaces on arbitrary line

Points on the line can be specified in the following way:

G1401 X_ Y_ A_ P_ T_ R_ Q_ (B_ C_ D_ E_)

where

X: X coordinate of the first point on the line.
Y: Y coordinate of the first point on the line.
A: Rotation angle of axis X. Default value is 0.
P: Number of holes including omit points.
T: Pitch space. Distance between two points.
R: Length of the line. If parameter T (pitch) is used, it should be empty.

Omit points: Set omit points by order of execution.

Omit points can be programmed by the following addresses:
- Omit point 1: B
- Omit point 2: C
- Omit point 3: D
- Omit point 4: E

Sample:
%O2002(Line)
M3 S1000 G91 G53

Start spindle speed specified by address S.
Programming of incremental data
Positioning in machine coordinate system.

G83 X10 Y10 Z-20 R-5 Q3 E0.5 P1 F100 L1

Setting drilling cycle data with execution of drilling.

G1401 X10 Y10 P5 T2 B1

Set line using same spaces starting from X10 Y10. Five holes are programmed; distance between holes is 2 mm. First point is skipped.

G80

Canned cycle cancel.

M5

Stop spindle

%
G1402 Hole pattern: Positions at different spaces on arbitrary line
Points on the line can be specified in the following way:

G1402 X_ Y_ A_ B_ C_ D_ E_ F_ H_ I_ J_ K_ R_

where

X: X coordinate of the first point on the line.
Y: Y coordinate of the first point on the line.
A: Rotation angle of axis X. Default value is 0.

Pitch: Distance between points should be set for every pitch. Maximum 11 points (ten pitches) can be programmed. Different pitches can be programmed by the following addresses (in the order of execution):

Pitch 1: B
Pitch 2: C
Pitch 3: D
Pitch 4: E
Pitch 5: F
Pitch 6: H
Pitch 7: I
Pitch 8: J
Pitch 9: K
Pitch 10: R

Sample:
%O2002(Line)

M3 S1000 G91 G53

G83 X10 Y10 Z-20 R-5 Q3 E0.5 P1 F100 L1 Setting drilling cycle data without execution of drilling.

G1402 X10 Y50 B2 C3 D4 Set line using different spaces starting from (X10 Y50). Three holes are programmed. Distances between the holes are 2-3-4 mms.

G80 Canned cycle cancel.
M5 Stop spindle
G1403 Hole Pattern: Positions of a grid

Points on the grid can be specified in the following way:

\[ \text{G1403 } \text{X} \_ \text{Y} \_ \text{U} \_ \text{V} \_ \text{I} \_ \text{J} \_ \text{K} \_ \text{R} \_ (\text{B} \_ \text{C} \_ \text{D} \_ \text{E} \_ ) \]

where

- **X**: Coordinate of axis X.
- **Y**: Coordinate of axis Y.
- **U**: Horizontal length of grid.
- **V**: Vertical length of grid.
- **I**: Number of holes on horizontal axis.
- **J**: Number of holes on vertical axis.
- **K**: Rotation angle of the horizontal axis based on axis X. Default value is 0.
- **R**: Angle between horizontal and vertical axes defined for the grid. Default value is 0.

**Omit points**: Set omit points by order of execution.

Omit points can be programmed by the following addresses:

- Omit point 1: B
- Omit point 2: C
- Omit point 3: D
- Omit point 4: E

Sample:

\%

\text{O2003(\text{Grid)}}

\text{M3 } \text{S1000} \text{ G91 G53}

Start spindle speed specified by address S.

Programming of incremental data.

Positioning in machine coordinate system.

\text{G83 X10 Y10 Z-20 R-5 Q3 E0.5 P1 F100 L1}

Setting drilling cycle data with execution of drilling.

\text{G1403 X10 Y10 U10 V10 I5 J3 K0 B1}

Set the grid starting from \((X10 \ Y10)\). Size is \((U, V)\) 5 holes are in a line, 3 holes in a column. First point is omitted.

\text{G80}

Canned cycle cancel.

\text{M5}

Stop spindle.

\%
G1404 Hole pattern: positions of a square
Data of square can be specified in the following way:

G1404 X_ Y_ U_ V_ I_ J_ K_ R_ (B_ C_ D_ E_ )

where

X: Coordinate of axis X.
Y: Coordinate of axis Y.
U: Horizontal length of rectangle.
V: Vertical length of rectangle.
I: Number of holes on horizontal axis.
J: Number of holes on vertical axis.
K: Rotation angle of the horizontal axis based on axis X. Default value is 0.
R: Angle between horizontal and vertical axes defined for the grid. Default value is 0.

Omit points: Set omit points by order of execution.
Omit points can be programmed by the following addresses:

Omit point 1: B
Omit point 2: C
Omit point 3: D
Omit point 4: E

Sample:
%O2004(Square)
M3 S1000 G91 G53
Start spindle speed specified by address S.
Programming of incremental data
Positioning in machine coordinate system.

G83 X10 Y10 Z-20 R-5 Q3 E0.5 P1 F100 L1
Setting drilling cycle data with execution of drilling.

G1404 X10 Y10 U10 V10 I5 J3 K0 B1
Setting square starting from (X10 Y10). Size of square is (U, V). Five horizontal three vertical holes will be drilled. First hole is omitted.

G80
Canned cycle cancel.
M5
Stop spindle.

%
G1405 Hole pattern: Positions of a circle
Holes on the circle can be specified in the following way:

G1405 X_ Y_ R_ A_ P_ (B_ C_ D_ E_)

where

X: Coordinate of center point X.
Y: Coordinate of center point Y.
R: Radius of circle
A: Angle between the line between center point of the circle and axis X. Default value is 0 (horizontal position)
P: Number of holes including points to be omitted.

Omit points: Set omit points by order of execution.
Omit points can be programmed by the following addresses:

- Omit point 1: B
- Omit point 2: C
- Omit point 3: D
- Omit point 4: E

Sample:

%O2005(Circle)

M3 S1000 G91 G53

Start spindle speed specified by address S.
Programming of incremental data
Positioning in machine coordinate system.

G83 X10 Y10 Z-20 R-5 Q3 E0.5 P1 F100 L1

Setting drilling cycle data with execution of drilling.

G1405 R10 X10 Y10 P15 A15 B1

Setting the circle. Center point is (X10 Y10), radius is, fifteen holes, first hole is shifted by 15 degree-. First point is omitted.

G80

Canned cycle cancel.

M5

Stop spindle.

%
G1406 Hole pattern: Positions of a same space arc
Arc can be programmed in the following way:

G1406 X_ Y_ R_ A_ T_ P_ (B_ C_ D_ E_)

where

X: Coordinate of center point X.
Y: Coordinate of center point Y.
R: Radius of circle
A: Angle between the line between center point of the circle and axis X. Default value is 0 (horizontal position)
T: Pitch angle. Angle between adjacent holes based on the center point.
P: Number of holes including points to be omitted.

Omit points: Set omit points by order of execution.
Omit points can be programmed by the following addresses:

- Omit point 1: B
- Omit point 2: C
- Omit point 3: D
- Omit point 4: E

Sample:
%O2006(Arc)
M3 S1000 G91 G53
G83 Z-20 R-5 Q3 E0.5 P1 F100 L0
G1406 R10 X10 Y10 P15 T1
G80
M5
%

Start spindle speed specified by address S.
Programming of incremental data
Positioning in machine coordinate system.
Setting drilling cycle data without execution of drilling.
Setting arc. Center point is (X10 Y10). Radius is 10 mm. Fifteen holes in every 15 degree.
Canned cycle cancel.
Stop spindle.
G1407 Hole pattern: Positions of a different space arc
Arc can be programmed in the following way:

G1407 X_ Y_ R_ A_ T_ P_ Q_ (B_ C_ . . .)

where

- **X:** Coordinate of center point X.
- **Y:** Coordinate of center point Y.
- **R:** Radius of circle
- **A:** Angle between the line between center point of the circle and axis X. Default value is 0 (horizontal position)

**Circle arc: different space**

**Pitch angle:** Angle of adjacent holes based on center point of the circle. Every pitch angle can be specified. Independently. Maximum 11 holes (10 angles) can be programmed.

Pitch angles can be specified on following addresses:

- Pitch angle 1: \( B \)
- Pitch angle 2: \( C \)
- Pitch angle 3: \( D \)
- Pitch angle 4: \( E \)
- Pitch angle 5: \( F \)
- Pitch angle 6: \( H \)
- Pitch angle 7: \( I \)
- Pitch angle 8: \( J \)
- Pitch angle 9: \( K \)
- Pitch angle 10: \( U \)

Sample:

```
%O2007(Arc)
M3 S1000 G91 G53
```

Start spindle speed specified by address S.
Programming of incremental data
Positioning in machine coordinate system.

```
G83 Z-20 R-5 Q3 E0.5 P1 F100 L0
G1407 R10 X10 Y10 B20 C5 D10
```

Setting drilling cycle data without execution of drilling.
Setting arc. Center point is \( (X10 \ Y10) \). Radius is 10 mm. Three holes are drilled, they pitch angles are 20-5-10 degree.

```
G80
M5
```

Canned cycle cancel.
Stop spindle.

%
Face milling

Square shape (G1410)
Face can be specified in the following way

\[ \text{G1410 } X_\_ Y_\_ W_\_ E_\_ D_\_ A_\_ Q_\_ R_\_ M_\_ P_\_ U_\_ V_\_ C_\_ T_\_ F_\_ Z_\_ B_\_ J_\_ H_\_ K_\_ \]

Cutting method
Uni and Bi direction

![Square shape: Uni and Bi](image)

Ring direction

![Square shape: ring](image)

**X:** X coordinate of center point of the square.

**Y:** Z coordinate of center point of the square.

**W:** Cutting method

Value can be:
1. Uni
2. Bi
3. Ring
**E:** Starting point of cutting.

Value can be:

1. Right bottom corner
2. Right top corner
3. Left bottom corner
4. Left top corner

**D:** Compensation cell (tool offset) of face milling tool (0-99).

**A:** Rotation angle of axis X. Default value is 0.

**Q:** Used only by Ring method. Size of frame on horizontal axis.

**R:** Used only by Ring method. Size of frame on vertical axis.

**M:** Approach gap. Distance between end of the tool and center point of the square before the cycle. Default value is 5.

**P:** Used only by Uni and Bi method. Abandon gap. Distance between end of the tool and center point of the square at the end of the cycle. Default value is 5.

**U:** Width of square (horizontal size)

**V:** Height of square (vertical size)

**C:** Depth of cut in tool radius direction. Value should be specified as percent of tool diameter (max. 70%).
T: T : Machining procedure
   1. Roughing
   2. Finishing

Procedure of roughing/finishing is different for uni/bi/ring method.

F: Feed.

Z: End point of the face in tool axis direction (bottom surface).

B: Depth of face in tool axis direction (cutting allowance).

J: Depth of cut in the tool axis direction. Used only by roughing. Roughing can be executed without filling this parameter.

H: Finishing allowance in the tool axis direction used by finishing procedure. Finishing can not be executed without this parameter

K: Amount of clearance in the tool axis direction. Default value is 3 mm.

Application of Parameters

| G1410 | D | K | T | Z | B | J | H | F | C | X | Y | U | V | Q | R | W | A | E | M | P |
| Roughing (Uni) | # & # # & & # & # # # # # # - - # & & & & & & |
| Finishing (Uni) | # & # # - - - # # # # # # # - - # & & & & & & |
| Roughing (Bi) | # & # # & & # & # # # # # # - - # & & & & & & |
| Finishing (Bi) | # & # # - - - # # # # # # # - - # & & & & & & |
| Roughing (Ring) | # & # # # & & # # & # # # # # # # # # & & & & & & |
| Finishing (Ring) | # & # # - - - # # # # # # # # # & & & & & & |

where:

#: It should be filled
&: It has default value
–: It does not used

Uni - method

![Diagram of Uni-method](image)
Roughing procedure

<1> Positioning above the start point A.

<2> The tool moves down to the position
    - bottom surface \(Z\),
    - + cutting allowance \(B\),
    - – depth of cut in the tool axis direction \(J\).

<3> The tool performs in-feed machining in the tool radius direction to point B.

<4> The tool elevates the distance of
    - depth of cut in the tool axis direction \(J\),
    - + clearance in the tool axis direction \(K\).

<5> Positioning in the tool radius direction to the next starting point C.

<6> The tool approaches the position of
    - depth of cut in the tool axis direction \(J\),
    - + clearance in the tool axis direction \(K\).

<7> Steps <3> - <6> are repeated until point D is reached.

<8> The tool elevates the distance of
    - bottom surface \(Z\),
    - + clearance in the tool axis direction \(K\),
    - + cutting allowance \(B\).

<9> Step <1> - <8> are repeated. Machining plane is decreased by
    - depth of cut in the tool axis direction \(J\),
    until
    - bottom surface \(Z\),
    - + finishing allowance \(H\)
    is reached.

<10> Tool retracts to
    - bottom surface \(Z\),
    - + clearance in the tool axis direction \(K\),
    - + cutting allowance in the tool axis direction \(B\).
**Finishing procedure**

<1> Positioning above the start point A.
<2> The tool moves down to the position bottom surface (Z).
<3> The tool performs in-feed machining in the tool radius direction to point B.
<4> The tool elevates the distance of clearance in the tool axis direction (K), + finishing allowance in the tool axis direction (H).
<5> Positioning in the tool radius direction to the next starting point C.
<6> The tool approaches in the tool axis direction the position of bottom surface (Z).
<7> Steps <3> - <6> are repeated until point D is reached.
<8> The tool elevates the distance of bottom surface (Z), + clearance in the tool axis direction (K), + cutting allowance (B).

**Bi - method**

![](image)

**Roughing procedure**

<1> Positioning above the start point A.
<2> The tool moves down to the position bottom surface (Z), + cutting allowance (B), – depth of cut in the tool axis direction (J).
<3> The tool performs in-feed machining in the tool radius direction to point B.
<4> Positioning along Y axis to the next start point C by the distance of depth of cut in tool radius direction (C).
The tool performs in-feed machining in the tool radius direction (point D).

Positioning along Y axis to the next start point by the distance of depth of cut in tool radius direction (C).

Steps <3> - <6> are repeated until the cutting allowance in the tool radius direction is removed (point E).

Retraction: Tool moves to
- clearance (K),
- cutting allowance (B),
- bottom surface in the tool axis direction (Z).

Steps <3> - <6> are repeated. Machining plane is decreased in the tool axis direction until bottom surface (Z),
- finishing allowance (H) is reached.

Tool retracts to
- clearance (K),
- cutting allowance (B),
- bottom surface in the tool axis direction (Z).

**Finishing procedure**

Positioning above the start point A.

The tool moves to the position of bottom surface in the tool axis direction (Z).

The tool performs in-feed machining in the tool radius direction (point B).

Positioning along Y axis to the next start point C by the distance of depth of cut in tool radius direction (C).

The tool performs in-feed machining in the tool radius direction (point D).

Positioning along Y axis to the next start point by the distance of depth of cut in tool radius direction (C).

Steps <3> - <6> are repeated until the cutting allowance in the tool radius direction is removed (point E).

Tool retracts to
- clearance (K),
- cutting allowance (B),
- bottom surface in the tool axis direction (Z).
**Ring method**

1. Positioning above the start point A.
2. The tool moves down to the position
   - bottom surface (Z),
   - + cutting allowance (B),
   - – depth of cut in the tool axis direction (J).
3. The tool performs in-feed machining on spiral path starting from approach gap (M).
   Offset between in-phase segments is depth of cut in tool radius direction (C).
4. Retraction: Tool moves to clearance (K),
   - + cutting allowance (B),
   - + bottom surface in the tool axis direction (Z).
5. Step <1> - <3> are repeated moving to -Z (tool axis) direction by depth of cut in the tool axis direction (J),
   until reaching end point (Z),
   - + finishing allowance in the tool axis direction (H).
6. Retraction: Tool moves to clearance (K),
   - + cutting allowance (B),
   - + bottom surface in the tool axis direction (Z).
Finishing procedure

<1> Positioning above the start point **A**.

<2> The tool moves down to the position of bottom surface in the tool axis direction (Z).

<3> The tool performs in-feed machining on spiral path starting from approach gap (M).

    Offset between in-phase segments is depth of cut in tool radius direction (C).

<4> Retraction: Tool moves to clearance (K),

    + cutting allowance (B),

    + bottom surface in the tool axis direction (Z).
Face milling circle surface (G1411)
Face can be specified in the following way

G1411 X_ Y_ W_ E_ D_ Q_ R_ M_ P_ C_ T_ F_ Z_ B_ J_ H_ K_

Cutting method
Uni and Bi direction

Circle shape: Uni and Bi direction

Ring direction

Circle shape: Ring direction

X: X coordinate of center point of the square.
Y: Z coordinate of center point of the square.
W: Cutting method

Value can be:
1. Uni
2. Bi
3. Ring

Uni direction Bi direction Ring direction
E: Starting point of cutting.

Value can be:
1. Right bottom corner
2. Right top corner
3. Left bottom corner
4. Left top corner

D: Compensation cell (tool offset) of face milling tool (0-99).

Q: Used only by ring method. Width of the ring.

R: Radius of the circle.

M: Approach gap. Distance between end of the tool and center point of the square before the cycle. Default value is 5 mm.

P: Used only by Uni and Bi methods. Abandon gap. Distance between and of the tool and center point of the square at the end of the cycle. Default value is 5 mm.

C: Depth of cut in tool radius direction. Value should be specified as percent of tool diameter (max. 70%).

T: T : Machining procedure
   1. Roughing
   2. Finishing

Procedure of roughing/finishing is different for uni/bi/ring method.

F: Feed.

Z: End point of the face in tool axis direction.

B: Depth of face in tool axis direction.

J: Depth of cut in tool axis direction. Used only by roughing. Roughing can be executed without filling this parameter.

H: Finishing allowance in tool axis direction used by finishing procedure. Finishing can not be executed without this parameter.

K: Amount of clearance in the tool axis direction.
Application of parameters

<table>
<thead>
<tr>
<th>G1411</th>
<th>D</th>
<th>K</th>
<th>T</th>
<th>Z</th>
<th>B</th>
<th>J</th>
<th>H</th>
<th>F</th>
<th>C</th>
<th>X</th>
<th>Y</th>
<th>R</th>
<th>Q</th>
<th>W</th>
<th>E</th>
<th>M</th>
<th>P</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roughing (Uni)</td>
<td>#</td>
<td>&amp;</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>&amp;</td>
<td>&amp;</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>-</td>
<td>#</td>
<td>&amp;</td>
<td>&amp;</td>
<td>&amp;</td>
<td></td>
</tr>
<tr>
<td>Finishing (Uni)</td>
<td>#</td>
<td>&amp;</td>
<td>#</td>
<td>#</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>-</td>
<td>#</td>
<td>&amp;</td>
<td>&amp;</td>
<td>&amp;</td>
<td></td>
</tr>
<tr>
<td>Roughing (Bi)</td>
<td>#</td>
<td>&amp;</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>&amp;</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>-</td>
<td>#</td>
<td>&amp;</td>
<td>&amp;</td>
<td>&amp;</td>
<td></td>
</tr>
<tr>
<td>Finishing (Bi)</td>
<td>#</td>
<td>&amp;</td>
<td>#</td>
<td>#</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>-</td>
<td>#</td>
<td>&amp;</td>
<td>&amp;</td>
<td>&amp;</td>
</tr>
<tr>
<td>Roughing (Gyűrű)</td>
<td>#</td>
<td>&amp;</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>&amp;</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>-</td>
<td>#</td>
<td>#</td>
<td>&amp;</td>
<td>&amp;</td>
<td>&amp;</td>
</tr>
<tr>
<td>Finishing (Gyűrű)</td>
<td>#</td>
<td>&amp;</td>
<td>#</td>
<td>#</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>#</td>
<td>-</td>
<td>#</td>
<td>&amp;</td>
<td>&amp;</td>
<td>&amp;</td>
</tr>
</tbody>
</table>

where:
- #: It should be filled
- &: It has default value
- –: It is not used

Uni - method

**Roughing procedure**

<1> Positioning above the start point A.

<2> The tool moves down to the position
   - bottom surface (Z),
   - + cutting allowance (B),
   - – depth of cut in the tool axis direction (J).

<3> The tool performs in-feed machining in the tool radius direction to point B.

<4> The tool elevates the distance of
   - depth of cut in the tool axis direction (J),
   - + clearance in the tool axis direction (K).

<5> Positioning in the tool radius direction to the next starting point C.

<6> The tool approaches the position of
   - depth of cut in the tool axis direction (J),
   - + clearance in the tool axis direction (K).
<7> Steps <3> - <6> are repeated until point D is reached.

<8> The tool elevates the distance of
bottom surface
+ clearance in the tool axis direction (Z),
+ cutting allowance (K),
+ finishing allowance (B).

<9> Step <1> - <8> are repeated. Machining plane is decreased by
depth of cut in the tool axis direction (J),
until
bottom surface (Z),
+ finishing allowance (H)
is reached.

<10> Tool retracts to
bottom surface (Z),
+ clearance in the tool axis direction (K),
+ cutting allowance in the tool axis direction (B).

**Finishing procedure**

<1> Positioning above the start point A.
<2> The tool moves down to the position
bottom surface (Z).
<3> The tool performs in-feed machining in the tool radius direction to point B.
<4> The tool elevates the distance of
clearance in the tool axis direction (K),
+ finishing allowance in the tool axis direction (H).
<5> Positioning in the tool radius direction to the next starting point C.
<6> The tool approaches in the tool axis direction the position of
bottom surface (Z).
<7> Steps <3> - <6> are repeated until point D is reached.
<8> The tool elevates the distance of
bottom surface (Z),
+ clearance in the tool axis direction (K),
+ cutting allowance (B).
Bi - method

Roughing procedure

<1> Positioning above the start point A.
<2> The tool moves down to the position
   bottom surface
   + cutting allowance (Z),
   – depth of cut in the tool axis direction (B),
<3> The tool performs in-feed machining in the tool radius direction to point B.
<4> Positioning along Y axis to the next start point C by the distance of
   depth of cut in tool radius direction (C).
<5> The tool performs in-feed machining in the tool radius direction (point D).
<6> Positioning along Y axis to the next start point by the distance of
   depth of cut in tool radius direction (C).
<7> Steps <3> - <6> are repeated until the cutting allowance in the tool radius direction is removed (point E).
<8> Retraction: Tool moves to
   clearance
   + cutting allowance (B),
   + bottom surface in the tool axis direction (Z).
<9> Step <1> - <8> are repeated. Machining plane is decreased in the tool axis direction until
   bottom surface
   + finishing allowance (H)
   is reached.
<10> Tool retracts to
    clearance
    + cutting allowance (B),
    + bottom surface in the tool axis direction (Z).
Finishing procedure

<1> Positioning above the start point A.
<2> The tool moves to the position of bottom surface in the tool axis direction (Z).
<3> The tool performs in-feed machining in the tool radius direction (point B).
<4> Positioning along Y axis to the next start point C by the distance of depth of cut in tool radius direction (C).
<5> The tool performs in-feed machining in the tool radius direction (point D).
<6> Positioning along Y axis to the next start point by the distance of depth of cut in tool radius direction (C).
<7> Steps <3> - <6> are repeated until the cutting allowance in the tool radius direction is removed (point E).
<8> Tool retracts to clearance (K), + cutting allowance (B), + bottom surface in the tool axis direction (Z).

Ring method

Roughing procedure

<1> Positioning above the start point A.
<2> The tool moves down to the position bottom surface (Z), + cutting allowance (B), depth of cut in the tool axis direction (J).
<3> The tool performs in-feed machining on spiral path starting from approach gap (M). Offset between in-phase segments is depth of cut in tool radius direction (C).

<4> Retraction: Tool moves to clearance (K), + cutting allowance (B), + bottom surface in the tool axis direction (Z).

<5> Step <1> - <3> are repeated moving to Z (tool axis) direction by depth of cut in the tool axis direction (J), until reaching end point (Z), + finishing allowance in the tool axis direction (H).

<6> Retraction: Tool moves to clearance (K), + cutting allowance (B), + bottom surface in the tool axis direction (Z).

**Finishing procedure**

<1> Positioning above the start point A.

<2> The tool moves down to the position of bottom surface in the tool axis direction (Z).

<3> The tool performs in-feed machining on spiral path starting from approach gap (M). Offset between in-phase segments is depth of cut in tool radius direction (C).

<4> Retraction: Tool moves to clearance (K), + cutting allowance (B), + bottom surface in the tool axis direction (Z).
Side cutting / Step milling

Square side (G1420)
Side cutting can be specified in the following way

G1420 M_ P_ X_ Y_ U_ V_ Q_ R_ Z_ B_ S_

where parameters specify the geometry of the step

- **M:** Cutting method
  - Value can be:
    1. External
    2. Internal

- **P:** Starting point of cutting.
  - Value can be:
    1. Right side
    2. Left side

- **X:** X coordinate of center point of the square.
- **Y:** Y coordinate of center point of the square.
- **U:** Width of square (horizontal size)
- **V:** Height of square (vertical size)
- **Q:** Type of the corner.
  - Value can be:
    1. Radius
    2. Chamfer
R: Value of radius/chamfer depending by the value of Q.
Z: Coordinate of end point of step in tool axis direction.
B: Depth of step in tool axis direction.
S: Width of step in tool radius direction.

**Block G1420 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1424-G1427 step milling technology blocks.**

**Circle side (G1421)**
Side cutting can be specified in the following way

\[ \text{G1421 M}_\_ \text{P}_\_ \text{X}_\_ \text{Y}_\_ \text{R}_\_ \text{Z}_\_ \text{B}_\_ \text{S}_\_ \]

where parameters specify the geometry of the step.

**M:** Cutting method

Value can be:
1. External
2. Internal

**P:** Starting point of cutting.

Value can be:
1. Right side
2. Left side

**X:** X coordinate of center point of the square.

**Y:** Y coordinate of center point of the square.

**R:** Radius of the circle.

**Z:** Coordinate of end point of step in tool axis direction.

**B:** Depth of step in tool axis direction.

**S:** Width of step in tool radius direction.
Block G1421 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1424-G1427 step milling technology blocks.

**Track side (G1422)**

Step can be specified in the following way

\[
\text{G1422 M_ P_ X_ Y_ U_ R_ Z_ B_ S_}
\]

where parameters specify the geometry of the step.

**M:** Cutting method

Value can be:
1. External
2. Internal

**P:** Starting point of cutting.

Value can be:
1. Right side
2. Left side

**X:** X coordinate of center point of the square.

**Y:** Y coordinate of center point of the square.

**U:** Distance between the center points of arcs.

**R:** Radius of arcs

**Z:** Coordinate of end point of step in tool axis direction.

**B:** Depth of step in tool axis direction.

**S:** Width of step in tool radius direction.

Block G1422 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1424-G1427 step milling technology blocks.
One side (G1423)
Step can be specified in the following way

\[ \text{G1423 } X \_ Y \_ U \_ M \_ P \_ Z \_ B \_ S \_ \]

where parameters specify the geometry of the step.

\begin{align*}
X &: \text{ X coordinate of center point of the square.} \\
Y &: \text{ Y coordinate of center point of the square.} \\
U &: \text{ Length of step in tool radius direction} \\
M &: \text{ Approach gap. Default value is 0mm.} \\
P &: \text{ Abandon gap. Default value is 0mm.} \\
Z &: \text{ Coordinate of end point of step in tool axis direction.} \\
B &: \text{ Depth of step in tool axis direction.} \\
S &: \text{ Width of step in tool radius direction.}
\end{align*}

**Block G1423 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1424-G1427 step milling technology blocks.**
Side milling technology (G1424)
Step can be specified in the following way

\[ \text{G1424 D} \_ \text{K} \_ \text{I} \_ \text{T} \_ \text{C} \_ \text{J} \_ \text{H} \_ \text{F} \_ \text{E} \_ \text{W} \_ \]

This block is based on the settings of G1420-G1423 geometry blocks

**D**: Compensation cell (tool offset) of face milling tool (0-99).

**K**: Approach gap. Distance between end of the tool and center point of the square before the cycle. Default value is 1 mm.

**I**: Depth of cut in tool radius direction.

**T**: Finishing allowance in tool radius direction.

**C**: Security plane in tool axis direction by executing approach and abandon movements. Default value is 2 mm.

**J**: Depth of cut in tool axis direction

**H**: Finishing allowance in tool axis direction

**F**: Feed in tool radius direction.

**E**: Feed in tool axis direction

**W**: Cutting direction (Spindle)

Value can be:
1. Same direction
2. Reverse direction
in the case of side cutting:

**Tool path**

**Outside**

1. Positioning above the start point A.
2. The tool moves down to the position bottom surface (Z), + cutting allowance (B), – depth of cut in the tool axis direction (J).
3. Tool moves in the direction of center point by feed. Movement is the sum of depth of cut in tool radius direction (I), + approach gap (K).
4. Machining procedure revolving around the center point by the step of depth of cut in tool radius direction (I), until removing finishing allowance in tool radius direction (T).
5. Positioning above the start point A.
<6> Tool descends in tool axis direction by the value of depth of cut in the tool axis direction \((J)\).

<7> Steps <4>-<6> are repeated until reaching coordinate of end point in tool axis direction + finishing allowance in tool axis direction \((Z), (H)\).

<8> Tool elevates to point R by the value of coordinate of end point in tool axis direction + cutting allowance in tool axis direction \((Z), (B)\).

Inside

Programming steps are similar for inside and outside cutting.

**Roughing**

Sample:

```plaintext
%O2020(Roughing)
M3 S1000 G90 G54
```

Start spindle speed specified by address S.

Programming of absolute data

Work coordinate system 1 selection

```plaintext
T1
G1420 M1 P1 X10 Y10 U30 V40 Q1 R4 Z-15 B10 S6
```

Square form definition. Center point stands at X10 Y10, size of square is defined as: width (X-size) 30 height (Y-size) 40, depth of step is from Z-5 to -10 deep, machining starts from right hand side, outside. There are radiuses (4 mm) at the corners.
G1424 D1 K2 I2.5 T0.5 C2 J1.5 H0.5 F150 E100 W1

Roughing of step. Diameter of the tool is read from compensation cell 1. Allowances are 0.5 mm. Depth of cut in tool radius direction is 2.5 mm, depth of cut in tool axis direction is 1.5 mm

Spindle stop

M5
%

Step (side cutting) bottom finishing technology (G1425)
Step can be specified in the following way

G1425 D_ K_ C_ I_ T_ F_ E_ W_

This block is based on the settings of G1420-G1423 geometry blocks

D: Compensation cell (tool offset) of face milling tool (0-99).
K: Approach gap in tool radius direction. Distance of center point of the tool from starting point of material removing. Default value is 1mm.
I: Depth of cut in tool radius direction in one round.
T: Finishing allowance in tool radius direction in one round.
C: Size of security lane in tool axis direction for approach and abandon movements. Default value is 2 mm.
F: Feed.
**E:** Feed in tool axis direction

**W** Cutting direction (Spindle)

Value can be:
1. Same direction
2. Reverse direction

In the case of side cutting:

**Tool path**

**Outside**

---

**Bottom finishing**

<1> Positioning above the start point A.

<2> The tool moves down to the position bottom surface
<3> Tool moves in the direction of center point by feed. Movement is the sum of depth of cut in tool radius direction (I), + approach gap (K).

<4> Machining procedure revolving around the center point by the step of depth of cut in tool radius direction (I), until removing finishing allowance in tool radius direction (T).

<5> Tool elevates to point R by the value of: coordinate of end point in tool axis direction (Z), + cutting allowance in tool axis direction (B).

Inside

Programming steps are similar for inside and outside cutting

**Bottom finishing**

Sample:
%O2021(Bottom finishing)
M3 S1000 G90 G54

Start spindle speed specified by address S.
Programming of absolute data
Work coordinate system 1 selection

T1 Tool for roughing is selected
G1420 M1 P1 X10 Y10 U30 V40 Q1 R4 Z-15 B10 S6

Square form definition. Center point stands at X10 Y10, size of square is defined as: width (X-size) 30 height (Y-size) 40, depth of step is from Z-5 to -10 deep, machining starts from right hand side, outside. There are radiuses (4 mm) at the corners.

G1425 D1 K2 I2.5 T0.5 C2 F150 E100 W1

Side cutting (step) bottom finishing. Diameter of the tool is read from compensation cell 1. Allowance is 0.5 mm in tool radius direction. Depth of cut in tool radius direction is 2.5 mm.

M5

Spindle stop

% Side cutting (step) side finishing technology (G1426)
Step can be specified in the following way

G1426 D_ K_ C_ J_ F_ W_

This block is based on the settings of G1420-G1423 geometry blocks

D: Compensation cell (tool offset) of face milling tool (0-99).
K: Approach and abandon gap in tool radius direction. Default value is 1 mm.
C: Size of security lane in tool axis direction for approach and abandon movements. Default value is 2 mm.

J: Depth of cut in tool axis direction in one round.

F: Feed.

E: Feed in tool axis direction

W Cutting direction (Spindle)

Value can be:
1. Same direction
2. Reverse direction

in the case of side cutting ("one"):

Tool path

Outside

--- Motion at rapid traverse rate (G00)
--- Linear motion at feed rate (G01)
Side finishing

<1> Positioning above the start point B.

<2> Tool moves in the tool axis direction to point R to the point of
    bottom surface \( (Z) \),
    + cutting allowance in tool axis direction \( (B) \),
    + security plane in tool axis direction \( (C) \).

<3> The tool moves down to the position
    bottom surface \( (Z) \),
    + cutting allowance in tool axis direction \( (B) \),
    – depth of cut in the tool axis direction \( (J) \).

<4> Approaching the surface by circle path.

<5> Removing
    finishing allowance in tool radius direction \( (T) \).

<6> Abandoning the surface by circle path.

<7> Retraction by
    security plane \( (C) \).

<8> Positioning above the start point B.

<9> Tool descends in tool axis direction by the value of
    depth of cut in the tool axis direction \( (J) \).

<10> Steps <4> - <8> are repeated until reaching
     coordinate of end point in tool axis direction \( (Z) \).

<11> Tool elevates to point R by the value of
     coordinate of end point in tool axis direction \( (Z) \),
     + cutting allowance in tool axis direction \( (B) \),
     + security plane \( (C) \).

Inside

Programming steps are similar for inside and outside cutting.
**Side finishing**

Sample:
%O2022(Side finishing)

M3 S1000 G90 G54

Start spindle speed specified by address S.

Programming of absolute data

Work coordinate system 1 selection

T1

G1420 M1 P1 X10 Y10 U30 V40 Q1 R4 Z-15 B10 S6

Square form definition. Center point stands at X10 Y10, size of square is defined as: width (X-size) 30 height (Y-size) 40, depth of step is from Z-5 to -10 deep, machining starts from right hand side, outside. There are radiuses (4 mm) at the corners.

G1426 D1 K2 C2 J1.5 F150 W1

Side finishing. Diameter of the tool is read from compensation cell 1. Allowance in tool radius direction is 0.5 mm. Depth of cut in tool axis direction is 1.5 mm

M5

Spindle stop

%Step (side cutting) chamfering technology (G1427)

Step can be specified in the following way

G1427 D_ K_ C_ I_ J_ T_ H_ F_ W_

This block is based on the settings of G1420-G1423 geometry blocks
**D:** Compensation cell (tool offset) of face milling tool (0-99).

**K:** Approach and abandon gap in tool radius direction. Default value is 1mm.

![Side finishing](image)

**C:** Security plane in tool axis direction by executing approach and abandon movements. Default value is 2 mm.

**I:** Size of chamfer

**J:** Angle of chamfering tool.

![During chamfering](image)

**T:** Small diameter of the tool.

**H:** Overhang of the chamfering tool.

![During chamfering](image)

**F:** Feed.

**W** Cutting direction (Spindle)

Value can be:
1. Same direction
2. Reverse direction

![Milling tool](image)
in the case of side cutting:

<1> Positioning above the start point B.
<2> Tool moves in the tool axis direction to the starting. Starting point is determined by
    coordinate of end point of step in tool axis direction (Z),
    depth of step in tool axis direction (B),
    security plane in tool axis direction (C),
    size of chamfer (I),
    small diameter of the tool (T),
    angle of chamfering tool (J),
    and overhang of the chamfering tool (H).

<3> Approaching the surface by circle path.
<4> Cutting the chamfer (I).
<5> Abandoning the surface by circle path.
Tool elevates to point R by the value of:

- coordinate of end point in tool axis direction \(Z\),
- cutting allowance in tool axis direction \(B\),
- security plane \(C\).

**Inside**

Programming steps are similar for inside and outside cutting.

**Chamfer**

Sample:

\%O2023(Chamfer)

M3 S1000 G90 G54

Start spindle speed specified by address S.

Programming of absolute data

T1

Work coordinate system 1 selection

Tool for chamfering is selected.

G1420 M1 P1 X10 Y10 U30 V40 Q1 R4 Z-15 B10 S6

Square form definition. Center point stands at X10 Y10, size of square is defined as: width (X-size) 30 height (Y-size) 40, depth of step is from Z-5 to -10 deep, machining starts from right hand side, outside. There are radiuses (4 mm) at the corners.

G1427 D1 K2 I3 T2.5 C2 J100 H0.3 F150 W1

Performing chamfer procedure. Diameter of the tool is read from compensation cell 1. Chamfer of 3 mm is performed by one round. Approach and abandon radius is 3 mm.

M5

Spindle stop

%
Pocket milling

Square pocket (G1430)

Pocket can be specified in the following way

G1430 X_ Y_ U_ V_ R_ Z_ B_

Block G1430 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1434-G1438 step milling technology blocks.

X:  X coordinate of center point of the square.
Y:  Y coordinate of center point of the square.
U:  Width of square (horizontal size)
V:  Height of square (vertical size)
R:  Value of radius at the corner.
Z:  Coordinate of end point of pocket in tool axis direction.
B:  Depth of pocket in tool axis direction.

Block G1430 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1434-G1438 step milling technology blocks.
Circle pocket (G1431)
Pocket can be specified in the following way

G1431 X_ Y_ R_ Z_ B_

Block G1431 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1434-G1438 step milling technology blocks.

X:  X coordinate of center point of the square.
Y:  Y coordinate of center point of the square.
R:  Radius of the circle
Z:  Coordinate of end point of pocket in tool axis direction.
B:  Depth of pocket in tool axis direction.

Block G1431 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1434-G1438 step milling technology blocks.
Track pocket (G1432)
Pocket can be specified in the following way

G1432 X_ Y_ U_ R_ Z_ B_

Block G1432 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1434-G1438 step milling technology blocks.

X: X coordinate of center point of the square.
Y: Y coordinate of center point of the square.
U: Distance between the center points of arcs.
R: Value of radius at the corner.
Z: Coordinate of end point of step in tool axis direction.
B: Depth of step in tool axis direction.

Block G1432 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1434-G1438 step milling technology blocks.
Groove pocket (G1433)
Pocket can be specified in the following way

\[ \text{G1433 \ X\ Y\ P\ U\ V\ K\ Q\ Z\ B} \]

Block G1433 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1434-G1438 step milling technology blocks.

X: X coordinate of center point of the square.
Y: Y coordinate of center point of the square.
P: Starting point of cutting.

Value can be:
1. Left side
2. Right side

U: Length of the groove in X direction.
V: Width of the groove in Y direction.
K: Approach gap in tool radius direction. Default value is 1 mm.
Q: Abandon gap in tool radius direction. Default value is 1 mm
Z: Coordinate of end point of pocket in tool axis direction.
B: Depth of step in tool axis direction.

Block G1433 is only for parameter definition. Values are stored in global variables #188-#199. Machining is performed by G1434-G1438 step milling technology blocks.
Pocket pre-drill technology (G1434)
Step can be specified in the following way

\[ \text{G1434 } S \_ Q \_ D \_ C \_ M \_ T \_ H \_ E \_ W \_ \]

where parameters specify the geometry of the step.

- **S:** Selection of drilling cycle. Value can be:
  1. G81 (Drilling spot boring cycle)
  2. G83 (Peck drilling cycle)
  3. G73 (High-speed peck drilling cycle)

- **Q:** It is the depth of the cut-in, in the cycles of G73 and G83.

- **D:** Compensation cell (tool offset) of face milling tool (0-99).

- **C:** Depth of cut in tool radius direction. Value should be specified as percent of tool diameter (max. 70%). This parameter determines drilling coordinates of pre drilling technology.

- **M:** Security plane in tool axis direction by executing approach and abandon movements. Default value is 2 mm.

- **H:** Finishing allowance in tool axis direction. This parameter determines drilling coordinates of pre drilling technology.

- **T:** Finishing allowance in tool radius direction. This parameter determines drilling coordinates of pre drilling technology.

- **E:** Feed in tool axis direction.

- **W:** Cutting direction (Spindle)

Value can be:
1. Same direction
2. Reverse direction
Roughing

Operations: According to the drilling cycles.

(Programming description: Drilling cycles)

Sample:

```
%O2030(Pre-drill)
M3 S1000 G90 G54

T1
G1430 X10 Y10 U30 V40 R4 Z-15 B10

G1434 S2 Q3 D1 C70 M2 T0.5 H0.5 E150 W1
```

Start spindle speed specified by address S.
Programming of absolute data
Work coordinate system 1 selection.

Tool for roughing is selected

Square form definition. Center point stands at X10 Y10, size of square is defined as:
width (X-size) 30 height (Y-size) 40, depth of step is from Z-5 to -10 deep. There are radiuses (4 mm) at the corners.

Pre-drilling pocket. Diameter of the tool is read from compensation cell 1. Drilling coordinates are defined by roughing parameters: finishing allowances are 0.5-0.5 mm. Depth of cut in tool axis direction is 3mm. End of the hole will be at (Z+H).

Spindle stop

%
Pocket roughing technology (G1435)
Step can be specified in the following way

\[ \text{G1435 D}_- \text{K}_- \text{S}_- \text{C}_- \text{M}_- \text{J}_- \text{H}_- \text{T}_- \text{F}_- \text{E}_- \text{W}_- \]

where parameters specify the geometry of the step.

- **D**: Compensation cell (tool offset) of face milling tool (0-99).
- **K**: Approach and abandon gap in tool radius direction. Default value is 1 mm.
- **S**: Pitch per revolution in tool axis direction for circular ramping. If no value is defined, linear ramping will be performed.
C: Depth of cut in tool radius direction. Value should be specified as percent of tool diameter (max. 70%).

M: Security plane in tool axis direction by executing approach and abandon movements. Default value is 2 mm.

J: Depth of cut in tool axis direction in one round. This parameter can be empty; roughing procedure can be performed without this.

H: Finishing allowance in tool axis direction. This parameter must be filled; roughing procedure can not be performed without filling this parameter!

T: Finishing allowance in tool radius direction in one round. This parameter must be filled; roughing procedure can not be performed without filling this parameter!

F: Feed.

E: Feed in tool axis direction.

W: Cutting direction (Spindle)

Value can be:
1. Same direction
2. Reverse direction

Tool path

Roughing

<1> Positioning above the start point.

<2> Positioning in tool axis direction to point R:
- coordinate of end point of pocket in tool axis direction (Z),
- + depth of pocket in tool axis direction (B),
- + security plane in tool axis direction (M).
Descending by feed in tool axis direction (E) to point specified by coordinate of end point of pocket in tool axis direction (Z) + depth of pocket in tool axis direction (B) – depth of cut in tool axis direction in one round (J).

Tool ramps circular by the radius of parameter (K) with feed in tool axis direction (E).

If parameter S is not empty to the point of coordinate of end point of pocket in tool axis direction (Z) + depth of pocket in tool axis direction (B) - depth of cut in tool axis direction in one round (J).

In-feed machining, tool moves farther away as it revolves around the starting point until reaching finishing allowance in tool radius direction (T) Depth of cut in tool radius direction (C) is applied.

Retracting by the value of Security plane in tool axis direction (M)

Positioning back to the starting point.

Descending by the value of depth of cut in tool axis direction (J).

Retraction in tool radius direction to the point R defined by coordinate of end point of pocket in tool axis direction (Z) + depth of pocket in tool axis direction (B) + security plane in tool axis direction (M)

Sample:

%O2031(Roughing)

M3 S1000 G90 G54

Start spindle speed specified by address S.

Programming of absolute data

Work coordinate system 1 selection.

T1 Tool for roughing is selected
G1430 X10 Y10 U30 V40 R4 Z-15 B10

Square form definition. Center point stands at X10 Y10, size of square is defined as: width (X-size) 30 height (Y-size) 40, depth of step is from Z-5 to -10 deep. There are radiiuses (4mm) at the corners.

G1435 D1 K2 S1.25 C70 M2 J1.5 T0.5 H0.5 F150 E100 W1

Pocket roughing. Diameter of the tool is read from compensation cell 1. Finishing allowances are 0.5-0.5 mm. Depth of cut in tool radius direction is 70 % of tool diameter. Depth of cut in tool axis direction is 1.5 mm. Tool ramps circular for approaching starting point. Tool descends 1.25 mm in every revolution.

M5

Spindle stop
Pocket bottom finishing technology (G1436)
Step can be specified in the following way

G1436 D K S C M T F E W

where parameters specify the geometry of the step.

D: Compensation cell (tool offset) of face milling tool (0-99).
K: Approach and abandon gap in tool radius direction. Default value is 1 mm.

S: Pitch per revolution in tool axis direction for circular ramping. If no value is defined, linear ramping will be performed.
C: Depth of cut in tool radius direction. Value should be specified as percent of tool diameter (max. 70%).

M: Security plane in tool axis direction by executing approach and abandon movements. Default value is 2 mm.

T: Finishing allowance in tool radius direction in one round. This parameter must be filled, roughing procedure can not be performed without filling this parameter!

F: Feed.

E: Feed in tool axis direction.

W: Cutting direction (Spindle)

Value can be:

1. Same direction
2. Reverse direction

![Diagram of milling tool showing same and reverse direction](image)

Tool path

![Diagram of tool path](image)

Bottom finishing

<1> Positioning above the start point.

<2> Positioning in tool axis direction to point R:

coordinate of end point of pocket in tool axis direction (Z),
+ depth of pocket in tool axis direction (B),
+ security plane in tool axis direction (M).
Descending by
feed in tool axis direction (E)
to point specified by
coordinate of end point of pocket in tool axis direction (Z)
+ depth of pocket in tool axis direction (B)
- depth of cut in tool axis direction in one round (J)
Tool ramps circular by
by the radius of parameter (K)
with feed in tool axis direction (E)
If parameter S is not empty to the point of
coordinate of end point of pocket in tool axis direction (Z)
+ depth of pocket in tool axis direction (B)
- depth of cut in tool axis direction in one round (J)
in-feed machining, tool moves farther away as it revolves around the starting point until reaching
finishing allowance in tool radius direction (T)
Depth of cut in tool radius direction (C)
is applied.

In-feed machining, tool moves farther away as it revolves around the starting point until reaching
finishing allowance in tool radius direction (T)
Depth of cut in tool radius direction (C)
is applied.

Positioning in tool axis direction to point R:
coordinate of end point of pocket in tool axis direction (Z)
+ depth of pocket in tool axis direction (B)
+ security plane in tool axis direction (M)

Sample:
%O2032(Bottom finishing)

M3 S1000 G90 G54

Start spindle speed specified by address S.
Programming of absolute data
Work coordinate system 1 selection.

T1

Tool for roughing is selected

G1430 X10 Y10 U30 V40 R4 Z-15 B10

Square form definition. Center point stands at X10 Y10, size of square is defined as: width (X-size) 30 height (Y-size) 40, depth of step is from Z-5 to -10 deep. There are radiuses (4 mm) at the corners.
Bottom finishing. Diameter of the tool is read from compensation cell 1. Finishing allowance is 0.5mm. Depth of cut in tool radius direction is 70% of tool diameter. Full depth is finished. Tool ramps circular for approaching starting point. Tool descends 1.5 mm in every revolution.

Spindle stop

Pocket side finishing technology (G1437)
Step can be specified in the following way

G1437 D_ K_ M_ J_ F_ W_

where parameters specify the geometry of the step.

D: Compensation cell (tool offset) of face milling tool (0-99).
K: Approach and abandon gap in tool radius direction. Default value is 1mm.
M: Security plane in tool axis direction by executing approach and abandon movements. Default value is 2 mm.
J: Depth of cut in tool axis direction in one round. This parameter can be empty; roughing procedure can be performed without this.
F: Feed.
**W:** Cutting direction (Spindle)

Value can be:
1. Same direction
2. Reverse direction

**Tool path**

- **Side finishing**
  - <1> Positioning above the start point.
  - <2> Positioning in tool axis direction to starting point defined by:
    - coordinate of end point of pocket in tool axis direction ($Z$)
    - depth of pocket in tool axis direction ($B$)
    - depth of cut in tool axis direction ($J$)
    - security plane in tool axis direction ($M$)
  - <3> Approaching in circle path
  - <4> Removing finishing allowance in tool radius direction. ($T$)
  - <5> Abandoning in circle path.
  - <6> Retraction by
    - security plane ($C$)
  - <7> Positioning to the start point.
  - <8> Steps <2>-<7> are repeated by the steps of
    - depth of cut in tool axis direction ($J$)
    - until reaching the value of
    - coordinate of end point of pocket in tool axis direction ($Z$)
Retraction in tool radius direction to the point R defined by
coordinate of end point of pocket in tool axis direction \((Z)\)
+ depth of pocket in tool axis direction \((B)\)
+ security plane in tool axis direction \((M)\)

Sample:

%O2033(Side finishing)
M3 S1000 G90 G54
Start spindle speed specified by address S.
Programming of absolute data
Work coordinate system 1 selection.

T1
tool for roughing is selected

G1430 X10 Y10 U30 V40 R4 Z-15 B10
Square form definition. Center point stands at X10 Y10, size of square is defined as: width (X-size) 30 height (Y-size) 40, depth of step is from Z-5 to -10 deep. There are radiuses (4 mm) at the corners.

G1437 D1 K2 M2 J1.5 F150 W1
Pocket side finishing. Diameter of the tool is read from compensation cell 1. Finishing allowance is 0.5 mm, depth of cut is 1.5 mm. Finishing allowance is fully removed

M5
Spindle stop

%
Pocket chamfering technology (G1438)
Step can be specified in the following way

G1438 D_ K_ M_ I_ J_ T_ H_ F_ W_

where parameters specify the geometry of the step.

D: Compensation cell (tool offset) of face milling tool (0-99).
K: Approach and abandon gap in tool radius direction. Default value is 1 mm.
M: Size of security plane in tool axis direction for approach and abandon movements. Default value is 2 mm.
I: Size of chamfer
J: Angle of chamfering tool.
T: Small diameter of the tool.
**H:** Overhang of the chamfering tool.

**F:** Feed.

**W:** Cutting direction (Spindle)

Value can be:
1. Same direction
2. Reverse direction

**Tool path**

---

**Chamfering**

1. Positioning above the start point.
2. Tool moves in the tool axis direction to the starting. Starting point is determined by coordinate of end point of step in tool axis direction (Z), depth of step in tool axis direction (B), security plane in tool axis direction (M), size of chamfer (I), small diameter of the tool (T), angle of chamfering tool (J) and overhang of the chamfering tool (H).
Approaching the surface by circle path.

Cutting the chamfer

Abandoning the surface by circle path.

Tool elevates to point R by the value of:
- coordinate of end point in tool axis direction (Z),
- + cutting allowance in tool axis direction (B),
- + security plane (M).

Sample:

%O2034(Chamfering)

M3 S1000 G90 G54
Start spindle speed specified by address S.
Programming of absolute data
Work coordinate system 1 selection.

T1 Tool for chamfering is selected

G1430 X10 Y10 U30 V40 R4 Z-15 B10
Square form definition. Center point stands at X10 Y10, size of square is defined as: width (X-size) 30 height (Y-size) 40, depth of step is from Z-5 to -10 deep, machining starts from right hand side, outside. There are radiuses (4 mm) at the corners.

G1438 D1 K2 M2 I1.5 J100 T2.5 H0.3 F150 W1
Performing chamfer procedure. Diameter of the tool is read from compensation cell 1. Chamfer of 1.5 mm is performed by one round. Approach and abandon radius is 2 mm.

M5 Spindle stop

%