The use of CNC-machines will still increase in the future.

Not only in industrial production also in small workshops conventional machines will be replaced by CNC-machines.

The application of CNC-technics is not bound to the classic machine tools such as lathes, milling machines or to the metalworking area. One could say, nearly every day a new application of CNC technics is realized. Practically all occupations such as technical designer, technical manager or salesman, skilled worker, methods engineer, controller, etc. will be confronted with CNC-technology in many ways.

CNC basic knowledge is important for everyone of them. How specialized this knowledge must be, will depend on the specific occupation.

EMCO MAIER & CO. is also producer of CNC production machines and since a long time experienced and active in technical education worldwide.

After producing the EMCO COMPACT 5 CNC which is used worldwide successfully for years, the EMCO F1-CNC has been developed. As the method and the concept of the EMCO COMPACT 5 CNC has been very successful, we designed the F1-CNC also that way: the student should work on the machine from the very first hour.
Many contents which are difficult to explain and often not understood when taught theoretically, can only be explained by working on the CNC-machine. Operating a CNC-machine, milling with different cutters, etc., can only be learned by practical working.

CNC milling machines are built in different types: horizontal, vertical, portal milling machines, machining centers, etc. Therefore we designed the machine so that CNC milling — covering all types — can be performed.

Consider machine and teaching material as a unit. The book BASIS is developed for the student. For the teacher an additional handbook and Overhead slides are available. Use this handbook in addition to the book BASIS if you want to do it in a self-teaching course.
1. General

- Technological data
- Finding the Chip Removal Values, Speeds
- Mounting the Tools
- Chucking the Workpieces
Technological data

1. Cutting speed (Vs)

\[ V_s (m/min) = \frac{d (mm) \times \pi \times S (rpm)}{1000} \]

- **Vs** = Cutting speed
- d = Diameter of workpiece
- S = Main spindle speed

The maximum cutting speed depends on:

- Material of workpiece:
  
  - The higher the resistance of the material, the lower the cutting speed.

The charts contain the following data:

- **Vs** = 44 m/min for aluminium (Torradur B)
- **Vs** = 35 m/min for soft steel
  
  - soft plastics
- **Vs** = 25 m/min for tool steel
  
  - hard plastics

- Material of tool:
  
  Carbide tools allow higher cutting speed than HSS tools.
  
  - values given in the charts are for HSS tools.

2. Spindle speed (S)

You calculate the speed of the milling spindle from cutting speed and diameter of milling cutter.

\[ S (rpm) = \frac{V_s (m/min) \times 1000}{d (mm) \times \pi} \]

3. Feed Rate and Depth of Cut

- **F** = Feed rate (mm/min)
- t = Depth of cut (mm)

Generally, feed rate and cutting speed depend on:

- workpiece material
- performance of machine and
- geometry of milling cutter.

Material of workpiece

The higher the material resistance the larger the feed and the depth of cut (limitation by milling cutter geometry).

The charts contain orientation values for the F1-CNC.

Connection F - t

The larger "t" the smaller "F" and vice versa.
Procedure

The technological data are written into the tool specification sheet.

Finding the feed rate and the depth of cut:

Material: aluminium

1. Depth of cut (t = 10 mm)  
2. Dia. of milling cutter (d=10 mm)

F = 60 mm/min

You can also proceed in a different way:

1. Feed rate F  
2. Dia. of milling cutter (10 mm)

Depth of cut 4.2 mm

Finding the speed of rotations:

- \( d \) Diameter of milling cutter
- \( V_s \) Correct cutting speed for the specific material
- \( S \) Spindle speed

The same procedure applies for drilling.

PS: Downcut milling - Conventional Milling

The specific knowledge is presupposed. However, with the F1-CNC the differences may be neglected.
Milling
Depth of cut · Cutter diameter · Feed

Drilling
Diameter of drill bit · Feed
Speed (of rotation) — Cutting speed — Feed

Attention:
When plunging in with cutter, halve feed values of mill chart.
Service and Maintenance of Machine

Lubrication:

Lubricate guideways of longitudinal, cross and vertical slide daily using oil gun (1 nipple on vertical slide, 2 nipples left side underneath longitudinal slide).

Pressure resistant, corrosion-protective oil with slip-stick reducing characteristics.
73 mm/sec (cSt) reference temperature 40° C.

E.g. CASTROL MAGNA BD 68
This corresponds to the CINCINNATI Specification P47.

Spindle taper for tool mounting

Interior taper of main spindle and tool taper have to be free of grease and dust (force locking).

Safety measures

Pay attention to the general and specific milling safety rules. The knowledge about them is pre-supposed.

Raw material

If you use aluminium, take only machinable aluminium.

Advisable material:

Torradur B, Al, Cu, Mg, PB F38, material no. 3.1645.51 according to DIN 1725/1747 or similar.

Tools

Use high quality and well sharpened tools only.
Clamping of Tools

Attention:
Spindle taper and tool taper must be dirt- and dust-free.

Clamping with collet chuck
Tools with cylindrical shaft are clamped with the collet chuck.

Clamping of tools

- Put collet into nut inclined so that the eccentric ring grips the groove of the collet. Screw nut with collet onto collet chuck.

Putting tool into collet and tighten nut with cylindrical pin in clockwise direction. For counter-holding of main spindle put cylindrical pin into collet holder.

Note:

Taking out the collet:
Unscrew nut. The eccentric ring in the nut presses the collet out when unscrewing.

Maintenance
Use oil and clean collet and collet chuck after use. Chips and dirt can damage the tapers and influence the precision.

Collets
You find the clamping capacity in inch and metric engraved on the collets. Diameters smaller or larger than indicated must not be clamped.
Clamping with shell end mill arbor

Using the arbor you can clamp tools up to a bore of 16 mm. The 4 spacing collars serve for adjusting the different width of the milling cutters.
Clamping Possibilities for Workpieces

Clamping bars
The clamping bars are mounted directly onto the slide depending on the relative workpiece.

Machine vice with stop
Width of jaw: 60 mm
Clamping capacity: 60 mm

Stepped clamping shoe
Height: 60 mm
For clamping a workpiece you need at least two clamping shoes.
**Intermediate plate**

To mount 3-jaw, 4-jaw chuck and independent. The intermediate plate itself is mounted on to the dividing attachment. The dividing attachment is clamped to the milling table with two T-nut screws.

---

**Adaptor plate**

To mount 3-jaw, 4-jaw chuck and independent. The adaptor plate itself is mounted on to the milling table.

---

**3-jaw chuck (2 x 3 Jaws)**

For holding of round, triangular and hexagonal workpieces centrically.

**4-jaw chuck (2 x 4 jaws)**

For holding of round, square and octogonal workpieces centrically.

**4-jaw independed chuck**

For holding of workpieces centrically and eccentri-cally.
The Dividing Attachment

Operating tips

TECHNICAL DATA

Diameter of rotary table: 150 mm
Worm reduction: 1:40
T-slots according to factory standard
Number of holes in dividing plates: 27, 33, 34, 36, 38, 39, 40, 42

OPERATING ELEMENTS

Clamping levers for rotary table (1):
Clamping levers are loosened during the dividing operation itself, but must be clamped before every machining operation.

Indexing pin with handle (2):
During direct dividing from 15° to 15°, the pin rests into the parameter notches of the rotary table. During indirect dividing (worm dividing) or free dividing by means of the graduated scale, the indexing pin must be pulled out and swivelled to the left.

The graduated scale (3) is for controlling the divisions.

Crank handle with index plunger (4) moves the worm which is engaged with the wormwheel of the rotary table during indirect dividing.

The shears serve to facilitate adding the number of holes when a fraction of a turn is to be added.

Disengaging and engaging the worm:
The allen head screw (5) is loosened. When the dividing plate is turned counterclockwise, the worm and wormwheel are disengaged. The rotary table can be turned by hand for direct indexing. By turning the dividing plate clockwise, worm and wormwheel are engaged. To facilitate engagement of worm and wormwheel, the rotary table should be moved slightly by hand.

The allen head screw (5) must again be retightened.
Types of Dividing

Indirect dividing:

Indirect dividing offers many more dividing possibilities and is more accurate because of the worm reduction of 1:40.

Indirect dividing method:

If the crank handle is turned 40 times, the rotary table makes 1 revolution (360°). With help of the dividing plates, exact fractions of turns can be executed.

Direct dividing:

Worm and wormwheel are disengaged.

Possibility 1:

Dividing by means of the indexing pin. Dividing possibility from 15° to 15° (i.e., maximum of 24 divisions within 360°).

Possibility 2:

The dividing can be done freely with the aid of the graduated scale on the rotary table.

Note

With indirect dividing the indexing pin is always disengaged. For manufacturing a workpiece the rotary table has to be fixed.

The indexing chart:

1st column: indicates number of divisions per 360°

2nd column: shows the corresponding angle of the division

3rd column: shows the number of 360° crank handle revolutions which are necessary

4th column: shows the number of holes to be added for each index plate
Example of an indirect dividing operation:

Desired division: 13 divisions in 360°

From the indexing table it can be seen that at the desired division 13, 3 full crank turns must be made plus a fraction turn of 3 additional holes on the indexing plate 39.

Practical execution:

1. The indexing plate with 39 holes is mounted.

2. In the indexing table one sees that at the division 13, 3 full turns plus 3 holes on the 39 plate have to be added. Therefore, the shears are fixed so that they include 4 holes.

3. The indexing plunger is placed in a hole of the 39 plate (marked black on the drawing) and the left shear arm moved until it touches the pin of the plunger.

4. Execution of the dividing operation: 3 full turns plus the fractional turn of the 3 added holes are made; that means that the plunger is placed in the black hole. One dividing operation is completed.

5. Next dividing operation:
The shears are turned until the left arm touches the pin again; the next dividing operation follows as described in 4. above.

NOTE: The shears may not be moved during the dividing operation, otherwise they do not serve their purpose as an orientation aid.

NOTE: If a larger number of holes has to be reached than the maximum opening of the shears allow, you have to set the difference of holes between the shears.

Example

21 divisions per 360° have to be carried through. From the chart one can see that one full turn plus the fractional turn of 38 holes on the disc 42 have to be carried through. 38 holes cannot be set. Thus: 42-38=4 holes. When dividing you make one additional turn (2 turn altogether) and turn back the difference of 4 holes (the shears comprise 5 holes).
### INDEX TABLE for MAXIMAT

Formula for the Calculation of the Hole Numbers Required

\[ z = \text{No. of divisions required for one revolution of the workpiece.} \]

\[ K = \text{No. of revolutions of handle for a complete revolution of the workpiece.} \]

\[ n = \text{No. of revolutions of handle for one dividing move: } n = \frac{K}{z} \]

Worm reduction of dividing head 1 : 40, i.e. \( K = 40 \).

<table>
<thead>
<tr>
<th>Division Desired</th>
<th>Degrees</th>
<th>No. of crank turns req'd</th>
<th>Amount of holes to be added for each index plate</th>
<th>Division Desired</th>
<th>Degrees</th>
<th>No. of crank turns req'd</th>
<th>Amount of holes to be added for each index plate</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>180°</td>
<td>20</td>
<td>32 33 34 36 38 40 42</td>
<td>2</td>
<td>180°</td>
<td>20</td>
<td>32 33 34 36 38 40 42</td>
</tr>
<tr>
<td>175°</td>
<td>19</td>
<td>12</td>
<td>33 34</td>
<td>175°</td>
<td>19</td>
<td>12</td>
<td>33 34</td>
</tr>
<tr>
<td>170°</td>
<td>18</td>
<td>24</td>
<td>35 36</td>
<td>170°</td>
<td>18</td>
<td>24</td>
<td>35 36</td>
</tr>
<tr>
<td>160°</td>
<td>17</td>
<td>21</td>
<td>36 38</td>
<td>160°</td>
<td>17</td>
<td>21</td>
<td>36 38</td>
</tr>
<tr>
<td>150°</td>
<td>16</td>
<td>18</td>
<td>38 40</td>
<td>150°</td>
<td>16</td>
<td>18</td>
<td>38 40</td>
</tr>
<tr>
<td>140°</td>
<td>15</td>
<td>15</td>
<td>40 42</td>
<td>140°</td>
<td>15</td>
<td>15</td>
<td>40 42</td>
</tr>
<tr>
<td>130°</td>
<td>14</td>
<td>12</td>
<td>42</td>
<td>130°</td>
<td>14</td>
<td>12</td>
<td>42</td>
</tr>
<tr>
<td>125°</td>
<td>13</td>
<td>24</td>
<td>1</td>
<td>125°</td>
<td>13</td>
<td>24</td>
<td>1</td>
</tr>
<tr>
<td>120°</td>
<td>12</td>
<td>9</td>
<td>2</td>
<td>120°</td>
<td>12</td>
<td>9</td>
<td>2</td>
</tr>
<tr>
<td>110°</td>
<td>11</td>
<td>3</td>
<td>3</td>
<td>110°</td>
<td>11</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>90°</td>
<td>10</td>
<td>24</td>
<td>4</td>
<td>90°</td>
<td>10</td>
<td>24</td>
<td>4</td>
</tr>
<tr>
<td>80°</td>
<td>8</td>
<td>24</td>
<td>5</td>
<td>80°</td>
<td>8</td>
<td>24</td>
<td>5</td>
</tr>
<tr>
<td>75°</td>
<td>8</td>
<td>9</td>
<td>6</td>
<td>75°</td>
<td>8</td>
<td>9</td>
<td>6</td>
</tr>
<tr>
<td>72°</td>
<td>8</td>
<td>12</td>
<td>12</td>
<td>72°</td>
<td>8</td>
<td>12</td>
<td>12</td>
</tr>
<tr>
<td>70°</td>
<td>7</td>
<td>21</td>
<td>13</td>
<td>70°</td>
<td>7</td>
<td>21</td>
<td>13</td>
</tr>
<tr>
<td>65°</td>
<td>7</td>
<td>6</td>
<td>14</td>
<td>65°</td>
<td>7</td>
<td>6</td>
<td>14</td>
</tr>
<tr>
<td>60°</td>
<td>6</td>
<td>18</td>
<td>15</td>
<td>60°</td>
<td>6</td>
<td>18</td>
<td>15</td>
</tr>
<tr>
<td>55°</td>
<td>6</td>
<td>3</td>
<td>16</td>
<td>55°</td>
<td>6</td>
<td>3</td>
<td>16</td>
</tr>
<tr>
<td>50°</td>
<td>5</td>
<td>12</td>
<td>17</td>
<td>50°</td>
<td>5</td>
<td>12</td>
<td>17</td>
</tr>
<tr>
<td>45°</td>
<td>4</td>
<td>12</td>
<td>18</td>
<td>45°</td>
<td>4</td>
<td>12</td>
<td>18</td>
</tr>
<tr>
<td>40°</td>
<td>4</td>
<td>16</td>
<td>19</td>
<td>40°</td>
<td>4</td>
<td>16</td>
<td>19</td>
</tr>
<tr>
<td>36°</td>
<td>4</td>
<td>24</td>
<td>20</td>
<td>36°</td>
<td>4</td>
<td>24</td>
<td>20</td>
</tr>
<tr>
<td>30°</td>
<td>3</td>
<td>9</td>
<td>21</td>
<td>30°</td>
<td>3</td>
<td>9</td>
<td>21</td>
</tr>
<tr>
<td>25°</td>
<td>3</td>
<td>12</td>
<td>22</td>
<td>25°</td>
<td>3</td>
<td>12</td>
<td>22</td>
</tr>
<tr>
<td>24°</td>
<td>2</td>
<td>21</td>
<td>23</td>
<td>24°</td>
<td>2</td>
<td>21</td>
<td>23</td>
</tr>
<tr>
<td>20°</td>
<td>2</td>
<td>18</td>
<td>24</td>
<td>20°</td>
<td>2</td>
<td>18</td>
<td>24</td>
</tr>
<tr>
<td>15°</td>
<td>2</td>
<td>18</td>
<td>25</td>
<td>15°</td>
<td>2</td>
<td>18</td>
<td>25</td>
</tr>
<tr>
<td>10°</td>
<td>2</td>
<td>4</td>
<td>26</td>
<td>10°</td>
<td>2</td>
<td>4</td>
<td>26</td>
</tr>
<tr>
<td>9°</td>
<td>2</td>
<td>8</td>
<td>27</td>
<td>9°</td>
<td>2</td>
<td>8</td>
<td>27</td>
</tr>
<tr>
<td>8°</td>
<td>2</td>
<td>4</td>
<td>28</td>
<td>8°</td>
<td>2</td>
<td>4</td>
<td>28</td>
</tr>
<tr>
<td>7°</td>
<td>2</td>
<td>12</td>
<td>29</td>
<td>7°</td>
<td>2</td>
<td>12</td>
<td>29</td>
</tr>
<tr>
<td>5°</td>
<td>2</td>
<td>12</td>
<td>30</td>
<td>5°</td>
<td>2</td>
<td>12</td>
<td>30</td>
</tr>
<tr>
<td>3°</td>
<td>2</td>
<td>12</td>
<td>31</td>
<td>3°</td>
<td>2</td>
<td>12</td>
<td>31</td>
</tr>
<tr>
<td>1°</td>
<td>2</td>
<td>12</td>
<td>32</td>
<td>1°</td>
<td>2</td>
<td>12</td>
<td>32</td>
</tr>
<tr>
<td>0°</td>
<td>1</td>
<td>12</td>
<td>33</td>
<td>0°</td>
<td>1</td>
<td>12</td>
<td>33</td>
</tr>
</tbody>
</table>

The table provides the amount of holes to be added for each index plate based on the division desired and the number of crank turns required.
Chapter 2: Handoperation

- Operating element (survey) 2.2
- Positioning of milling cutter 2.4
- Traverse indication 2.7
- Input of X, Y, Z values 2.8
- Switching feed motors "Curventless“ 2.11
Traverse — Hand Operation

Display

After switching on the machine, the figure 0 appears. Lamps X, Y or Z are on.

If you traverse in -X, the lamp X lights up. When you take your finger from the key, the traverse distance is shown in 1/100 mm on the VDU. With a distance of 2.45 the display indicates 245.

If you press the Z-key, the light jumps to the Z-lamp. After you lift your finger from the key, the traverse distance appears (with 6.28 mm 628 will appear)

Monitor

The screen shows zero for X, Y, Z when you switch it on.

With the exception of rapid traverse the indication is shown continuously in steps of 0.5 mm.

Minus sign on display

With the exception of rapid traverse the indication is shown continuously in steps of 0.5 mm.
Input of X, Y, Z Zero-Values from any chosen Milling Position

The display should indicate zero, in case the milling cutter stands at a given point (X=0, Y=0, Z=0).

You can program the X,Y,Z displays to indicate zero.

The milling cutter is at a distance of 22,15 mm to the workpiece edge in X. The display indicates whatever value.

In case the milling cutter traverses in +X direction by 22,15 mm, then the display should indicate the value X=0.

Procedure:

1. The lamp X on the display lights up
2. Press INP - the lamp X flashes
3. Put in the value 2215 (no plus/minus sign, because the milling cutter should indicate with plus "traverse direction 0").
4. Press key INP. The flashing of the X-lamp stops.

You can enter the Y,Z values in the same way.

When programming minus-values first put in the figures, then press key minus.
Application of Path Programming in Hand Operation Mode

Zero point for the dimensioning is the workpiece edge. The milling cutter shall move to this point. The displays shall be set zero.

Procedure:

1. Scratch surface, set Z-display zero.

2. Scratch surface in X-direction. Put in value of milling cutter radius $r$.


Note:

You can traverse after scratching as you like. If you program the zero-point, you have to add to the X,Y display the radius value and put it in.

Exercise:

1. Program the display $X,Y,Z=0$ if the milling cutter is positioned onto the edge.

2. Move the milling cutter to the indicated position.
Switching Feed Motors "Currentless"

When switching on the machine the feed motors are currentless.
If you have - in hand- or CNC-operation mode - moved the slides the feed motors stay under power.

Switching currentless - with no program being stored

1. Switch to CNC-operation mode: Press \( \text{H/C} \) key.
2. Press key \( \rightarrow \). The light jumps to \( G \).
3. Key in \( 64 \). The number appears on the VDU.
4. Press key \( \text{INP} \). Now the feed motors are switched currentless.

Switching currentless - with a program being stored

G64 is a pure switching function. It is not stored.

1. Press key \( \rightarrow \) so that G light gets on.
2. When a number appears on the VDU, press \( \text{DEL} \).
3. Key in \( 64 \).
4. Press key \( \text{INP} \). Now the feed motors are switched currentless.
Operating Elements
Control Elements
Hand Operation

1. Main switch
   Turn key to the right. Machine and control part are under power (except emergency stop button is pressed).

2. Control lamp main switch
   When main switch is on, lamp is on.

3. Emergency stop button
   Control unit, feed motors and main motor are cut off from power by pressing emergency stop button: turn button to the left - it will jump back to original position. Main switch has to be switched on again.
4. Switch for main spindle

Turn switch to the right.

5. Turning knob for speed control of main spindle

6. Ammeter

Shows power consumption of main spindle motor. In order to protect motor against overload, the power consumption should not surpass 2 A with 220-240 V or 4 A with 100-110 V.

7. Feed keys for longitudinal, cross and vertical slide

8. Rapid traverse key

If keys for feed and rapid traverse are pressed together, then the relative slide will move with rapid traverse speed.

9. Turning knob for setting the feed rate

10. Inch/metric switch and switch for changing the axis system

11. Digital read-out for slide movement

± X, ± Y, ± Z are shown in 1/100 mm or 1/1000 inch.

Plus movement without sign
Minus movement by a light beam

- 125

X -1.25 mm or -0.125 inch

12. Control lamp for hand operation

13. H/C switch key: hand operation/CNC operation

If you press the H/C key the light of the control lamp hand operation will jump to CNC operation (operation mode: CNC). By pressing the key once again the light will jump back (operation mode: hand operation).

14. DEL key

The X, Y, Z values are set to zero.

15. The → key

With the → key you can switch from X to Y to Z without movement of slides.

16. The [ key

With the [ key you enter the values for slide movements.

17. M-key

Activates switching exits.
1. Scratching front sides and top side

With milling most measurements refer to outer edges. In order to use the measurements of the technical drawing you have to "zero-set" the display and use as reference/starting point the outer edges.

Example

Milling cutter with dia. 10 mm.

Move milling cutter in Z-direction until you scratch surface slightly.

Set Z-display to zero (press key DEL).

- Scratch front side in X-direction.
- Set X-display to zero (press key DEL)

- Scratch front side in Y-direction.
- Set Y-display to zero (press key DEL)
2. Zero-setting of Display to Zero Point of Dimensioning (Example: Milling)

Example: Milling of groove
- The groove is milled using a 8 mm cutter.
- Zero point for the dimensioning is the workpiece edge and surface.
- The measures refer to the center of the milling cutter.

Consequence
Move axis of milling cutter to edge of workpiece.

a) Scratching of all 3 surfaces and zero-setting of X,Y,Z.

b) Move by value of milling cutter radius into X-direction. Set X to "zero".
Exercise

Move milling cutter such that all display values are at "zero".

Exercise

Mill a recess as in drawing. Enter the following values:

<table>
<thead>
<tr>
<th>Spindle speed</th>
<th>S (rpm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Feed mm/min</td>
<td></td>
</tr>
<tr>
<td>Infeed in X (mm)</td>
<td></td>
</tr>
<tr>
<td>Infeed in Z (mm)</td>
<td></td>
</tr>
</tbody>
</table>

Pay attention to set correct feed.
Chapter 3
CNC-Operation – Survey

- Operating and control elements 3.2 – 3.3
- Preparatory functions, miscellaneous-/Switching functions 3.4 – 3.5
- Alarm signs 3.6
- Possible inputs 3.7
- Operation CNC
  Operation magnetic tape 3.9
Operating Elements
Control Elements
CNC-Operation

1. Main switch with removable key. Memory is being cleared when switching off.

2. Control lamp shows the power supply of machine and control unit.

3. Emergency stop button with interlock. Unlocking of button: turn button to the left. To switch on machine, turn main switch to zero and to 1 again. When switching off also memory will be cleared.

4. Optional switch for axis system and for metric or inch mode of operation.
5. Switch for main spindle

Position 1 (main spindle ON, without M03)

Position CNC: main spindle is switched on by programming M03 and switched off by M05, M06 (with F*0) and M30.

6. Ammeter

7. Magnetic tape

8. H/C switch key
   Manual/CNC operation

9. Control lamp CNC operation

10. START key
    The program is being worked off

11. Keys for program input, correction, storing of program on tape, V24 operation etc. (see detailed explanations)

11.1. Number keys [0 - 9]

11.2. The minus sign key
   To enter minus values the minus sign [−] has to be pressed after input of numbers.

11.3. INPUT key (INPUT = storing)
   Storing key

11.4. DELETE key (DELETE = erase)
   Erasing key

11.5. FORWARD key (FORWARD)
   Program jumps forward block by block

11.6. REVERSE key (REVERSE)
   Program jumps backwards block by block

11.7. Arrow key
   Display jumps word by word

11.8. M key: key for entering of miscellaneous functions.

12. VDU (display):
   Indicates values for address letters and modes of operation

13. Control lamp address letters

14. Control of milling spindle speed
CNC-Operation (Survey)

Survey
Preparatory Functions, G-Codes

G00 Rapid traverse
V: N3/G00/X±5/Y±4/Z±5
H: N3/G00/X±4/Y±5/Z±5

G01 Linear interpolation
V: N3/G01/X±5/Y±4/Z±5/F3
H: N3/G01/X±4/Y±5/Z±5/F3

G02 Circular interpolation clockwise
G03 Circular interpolation counterclockwise
Quadrants:
V: N3/G02/X±5/Y±4/Z±5/F3
N3/G03/X±4/Y±5/Z±5/F3
H: N3/G02/X±4/Y±5/Z±5/F3
N3/M99/J2/K2 (Partial circles)

G04 Dwell
N3/G04

G21 Empty block
N3/G21

G25 Sub-routine program call
N3/G25/L(F)3

G27 Jump instruction
N3/G27/L(F)3

G40 Tool radius compensation cancelled
N3/G40

G45 Add tool radius
N3/G45

G46 Subtract tool radius
N3/G46

G47 Add tool radius twice
N3/G47

G48 Subtract tool radius twice
N3/G48

G64 Feed motors without current
(switching function)
N3/G64

G65 Magnetic tape operation
(switching function)
N3/G65

G66 Activating RS 232 Interface
N3/G66

G72 Pocket milling cycle
H: N3/G72/X±4/Y±5

G74 Thread-cutting cycle
(left-hand)
N3/G74/K3/Z±5/F3

G81 Fixed boring cycle
N3/G81/Z±5/F3

G82 Fixed boring cycle with dwell
N3/G82/Z±5/F3

G83 Fixed boring cycle with chip removal
N3/G83/Z±5/F3
CNC-Operation (Survey)

G84 Thread-cutting cycle
   N3/G84/K3/Z+5/F3

G85 Fixed reaming cycle
   N3/G85/Z+5/F3

G89 Fixed reaming cycle with dwell
   N3/G89/Z+5/F3

G90 Absolute value programming
   N3/G90

G91 Incremental value programming
   N3/G91

G92 Offset of reference point

V = Vertical
H = Horizontal

Miscellaneous or Switching Functions

M00 - Dwell
   N3/M00

M03 - Milling spindle ON, clockwise
   N3/M03

M05 - Milling spindle OFF
   N3/M05

M06 - Tool offset, milling cutter radius input
   N3/M06/D5/S4/Z±5/T3

M17 - Return to main program
   N3/M17

M08

M09

M20 - Switching exits
   N3/M21

M22

M23

M26 - Switching exit - impulse
   N3/M26/H3

M30 - Program end
   N3/M30

M99 - Parameters circular interpolation
   (in connection with G02/03)
   N3/M99/J3/K3
Alarm Signs

A00: Wrong G/M code
A01: Wrong radius / M99
A02: Wrong Z-value
A03: Wrong F-value
A04: Wrong Z-value
A05: M30 code missing
A06: M03 code missing
A07: No significance
A08: Tape end with cassette operation
       SAVE
A09: Program not found
A10: Writing protection
A11: Loading mistake
A12: Checking mistake
A13: Inch/mm switching with full program memory
A14: Wrong mill head position/path increment with LOAD \[\] /M or \[\] /M
A15: Wrong Y-value
A16: Value of milling cutter radius missing
A17: Wrong sub-routine
A18: Path milling cutter compensation smaller zero
Possible Inputs  
(Otherwise alarm signs)

<table>
<thead>
<tr>
<th></th>
<th>Metric Values</th>
<th>Unit (mm)</th>
<th>Inch Values</th>
<th>Unit (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$X_V$</td>
<td>0-19999</td>
<td>1/100 mm</td>
<td>0-7999</td>
<td>1/1000&quot;</td>
</tr>
<tr>
<td>$X_H$</td>
<td>0-9999</td>
<td>1/100 mm</td>
<td>0-3999</td>
<td>1/1000&quot;</td>
</tr>
<tr>
<td>$Y_V$</td>
<td>0-9999</td>
<td>1/100 mm</td>
<td>0-3999</td>
<td>1/1000&quot;</td>
</tr>
<tr>
<td>$Y_H$</td>
<td>0-19999</td>
<td>1/100 mm</td>
<td>0-7999</td>
<td>1/1000&quot;</td>
</tr>
<tr>
<td>$Z_{VH}$</td>
<td>0-19999</td>
<td>1/100 mm</td>
<td>0-7999</td>
<td>1/1000&quot;</td>
</tr>
<tr>
<td>Radii</td>
<td>0-9999</td>
<td>1/100 mm</td>
<td>0-3999</td>
<td>1/1000&quot;</td>
</tr>
<tr>
<td>D(X) milling cutter radius with M06</td>
<td>0-9999</td>
<td>1/100 mm</td>
<td>0-3999</td>
<td>1/1000&quot;</td>
</tr>
<tr>
<td>$F$</td>
<td>2-499</td>
<td>mm/min</td>
<td>2-199</td>
<td>1/10&quot;/min</td>
</tr>
<tr>
<td>T(F) tool address M06</td>
<td>0-499</td>
<td>1</td>
<td>0-199</td>
<td>1</td>
</tr>
<tr>
<td>L(F) jump instructions</td>
<td>0-221</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>H(F) exit signs M26</td>
<td>0-299</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>J/K circular parameter</td>
<td>0-90</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Adresses

Operation CNC

**INP** Storing of word contents
**DEL** Deleting of word contents
**FWD** Forward in program block by block
**REV** Backward in program block by block
**→** Forward in block word by word
**M** Input of M-functions

Program hold:
**INP** + **FWD**

Program interruption
**INP** + **REV**

Delete program
**DEL** + **INP**
First **DEL** then **INP**
**DEL** remains pressed.

Delete alarm
**INP** + **REV**

Insert block

Delete block
**~** + **DEL**

Single block mode
1 2 3 etc. + **START**

Testrun:
**M**

Operation - Magnetic tape

Storing of program on tape
G65 **INP** → **INP** → **FWD** → Put in program number → **INP**

Transmit program from tape to memory
G65 **INP** → **INP** → Select program number → **INP**

Delete tape contents
G65 **INP**

↓

→ + **DEL**
# Chapter 4

## CNC-Basics

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>CNC-lathe – The control</td>
<td>4.1</td>
</tr>
<tr>
<td>CNC-machine – Main elements</td>
<td>4.2 – 4.3</td>
</tr>
<tr>
<td>What happens in CNC-manufacture</td>
<td>4.4 – 4.7</td>
</tr>
<tr>
<td>Differences in manufacture using a handoperated or a CNC-machine</td>
<td>4.8 – 4.9</td>
</tr>
<tr>
<td>This you are going to learn</td>
<td>4.11</td>
</tr>
<tr>
<td>What is programming</td>
<td>4.13 – 4.15</td>
</tr>
<tr>
<td>The coding standards</td>
<td>4.17 – 4.19</td>
</tr>
<tr>
<td>Program structure</td>
<td>4.21 – 4.23</td>
</tr>
<tr>
<td>G00/G01</td>
<td>4.25</td>
</tr>
<tr>
<td>Description of path lengths for slide movements</td>
<td>4.27</td>
</tr>
<tr>
<td>The CNC-program (structure)</td>
<td>4.29</td>
</tr>
<tr>
<td>The address words of the program sheet F1-CNC</td>
<td>4.31 – 4.33</td>
</tr>
<tr>
<td>Standardization of axis systems for CNC-machines</td>
<td>4.35 – 4.41</td>
</tr>
<tr>
<td>Concept of programming – Methods of programming</td>
<td>4.43</td>
</tr>
<tr>
<td>Dimensions of drawings</td>
<td>4.45</td>
</tr>
<tr>
<td>The modes of programming</td>
<td>4.47</td>
</tr>
<tr>
<td>G90/G91</td>
<td>4.49 – 4.51</td>
</tr>
<tr>
<td>Topic</td>
<td>Page(s)</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>-----------</td>
</tr>
<tr>
<td>Determining the coordinates for programming in absolute mode</td>
<td>4.53 – 4.55d</td>
</tr>
<tr>
<td>Information to the control concerning the workpiece zero-point</td>
<td>4.57</td>
</tr>
<tr>
<td>Fixing the origin of the coordinates on the F1-CNC (workpiece zero-point)</td>
<td>4.59</td>
</tr>
<tr>
<td>Fixing the zero-point of coordinates with G92 – Programmed offset of reference point</td>
<td>4.61 – 4.69</td>
</tr>
<tr>
<td>Various workpiece zero-points in one program</td>
<td>4.71 – 4.73</td>
</tr>
<tr>
<td>Mixed programming</td>
<td>4.75 – 4.77</td>
</tr>
<tr>
<td>Connection: G92 – Zero-point offset/ M06 – Tool lengths compensation</td>
<td>4.79</td>
</tr>
<tr>
<td>Some tips for procedure</td>
<td>4.81 – 4.83</td>
</tr>
<tr>
<td>The M-functions</td>
<td>4.85 – 4.87</td>
</tr>
<tr>
<td>Description of block formats</td>
<td>4.89</td>
</tr>
<tr>
<td>Types of controls of CNC machine tools</td>
<td>4.91 – 4.97</td>
</tr>
<tr>
<td>Programming – Geometry</td>
<td>4.99 – 4.143</td>
</tr>
</tbody>
</table>
What is a CNC-lathe?
- A machine which we feed with figures and letters = DATA INPUT
- A machine which "understands" the data which processes it and calculates. = DATA PROCESSING
- A machine which passes on this calculated data in form of instructions. = EXECUTION
- A machine which follows the instruction

Meanings in daily use
The meanings change quite often in their daily use. NC-machines were originally machines with numerical control, but no microprocessor. Today such machines are obsolete. The program was read in directly from the perforated tape.

Today NC-machines comprise all types CNC, DNC or AC types.
CNC — Machine — Main Elements —
A "humanized" Comparrison

Central Processing Unit = Microprocessor. Let’s call it the director. He delegates, takes decision, calculates. A watch gives him the feeling for time, but he does not have any specialist knowledge.

Memory = RAM Remembers the program

Operating program (EPROM) = specialists. They know everything.

Data Input: Via keys or magnetic tape

Data on digital read-out

Input element: can be compared to a secretary

Output element: Let’s call him press speaker.

Output element (Interface): Chief operator. He receives orders and passes them on.

Machine

Amplifier (foreman)
CNC-Machine – Main Elements

Data Input

Operating program = EPROMS (Specialists)

Interface element (secretary)

Central processing unit = Microprocessor (Director)

Memory = RAM

Output element = Amplifier (foreman)

Output element (press speaker)

Digital read-out

CNC-machine – Main elements
What happens in CNC-Manufacture

In the computer nothing happens without the director. There is a strict hierarchy.

What happens if you press the key START?

1. Secretary —> Director:
"They pressed START!"
Director asks memory:
"Did they put in program end M30?"
If yes, the program can start.

2. Director —> Specialists:
We want to machine a groove in a certain angle.

3. Specialist —> Director:
"Yes, o.k."

4. Director —> Memory:
"Please give me the data!"

5. Memory —> Director:
X,Y slides have to be moved in ratio 1 : 4.

6. Director calculates and gives data to chief operator. With the aid of the watch he also determines the operating speed (when threading he waits for the main spindle position).

7. Chief operator —> Foreman:
Move X slide with feed size F1 and Y slide with feed size F2.

8. Director —> Press speaker:
"The block is finished. We work on the next. Let them know!"
What happens in CNC-Manufacture?

Data Input
Interface element (secretary)
Operating program = EPROMS (Specialists)
CNC-Machine

Digital read-out
Output element (press speaker)
Memory = RAM
Amplifier (foreman)

Central processing unit = Microprocessor (Director)

Output element = Interface (Chief operator)
What happens in CNC-Manufacture

What knowledge is necessary in order to manufacture, using a hand operated or a CNC lathe?

Hand operated machine

NC-machine
Differences in Manufacture using a hand operated or a CNC-Machine (Survey)

Hand operated machine

CNC-Machine

Necessary information

Technical drawing

Necessary means

<table>
<thead>
<tr>
<th></th>
<th>Lathe</th>
<th>Chucking devices</th>
<th>Tools</th>
</tr>
</thead>
<tbody>
<tr>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

Necessary knowledge/Capabilities (to execute operation)

<table>
<thead>
<tr>
<th></th>
<th>Reading of technical drawings</th>
<th>Knowledge about tool geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

CNC machine – hand operated machine
Differences in manufacture, using a hand operated or a CNC-machine – continued

<table>
<thead>
<tr>
<th>Hand operated machine</th>
<th>NC-machine</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Technological information</strong></td>
<td></td>
</tr>
<tr>
<td>+ Cutting speed depending on</td>
<td></td>
</tr>
<tr>
<td>- material of workpiece</td>
<td></td>
</tr>
<tr>
<td>- tool (HSS, carbide tipped)</td>
<td></td>
</tr>
<tr>
<td>- type of operation</td>
<td></td>
</tr>
<tr>
<td>+ Feed rate</td>
<td></td>
</tr>
<tr>
<td>+ Cutting depth</td>
<td></td>
</tr>
<tr>
<td>+ Performance and dimensions of machine</td>
<td></td>
</tr>
</tbody>
</table>

**Execution**

- Operator must know how to control the machine
- + Writing the NC-program
- + Input of NC-program
  + Writing the NC-program
  + Input of NC-program
- + Preparing the machine
- + Execution
This you are going to learn
A rough survey

Set up a CNC-program

Enter all informations into program sheet.
Rules how to write these data have to be learned.

Put in program

You have to put in the information into the control. The control stores the information. You have to follow certain rules.

Give instruction to manufacture

The control works with the information entered - it calculates and gives instructions to the machine tool.

Check result
Correct program
Improve (optimize) program.
What is Programming?

Programming means to feed the computer with such data which it understands.

In other words, we have to "spoon-feed" the computer, list the data in orderly sequence and in a language which is familiar to the computer, which it understands, so that it can process the information.

The operator does not understand the Chinese commands, because he does not speak this language.

The CNC-machine does not understand the human language.

We have to feed the CNC-machine with data in a language it will understand. This language is "encoded".
Do you already know programming?

If you have operated a machine tool you automatically carried out the right movements.

Your brain gave instruction to your hands to operate the switches and levers in the correct sequence.

This job was automated to a large extent.

When programming you have to write down all instructions.

The instructions and informations must be

- in a systematic sequence
- complete
- and accurate.

They are given to the CNC-machine in a coded form.
The Coding Standards

The program structure for numerically controlled machine tools:

How to code informations and instructions is defined by standards.

The standards are:

- Program structure for numerically controlled machine tools.

- According to DIN 66025 (German Industrial Standards)


MOVE LONGITUDINAL SLIDE 10mm TO THE LEFT 200mm/min

N... /G01 /x+10 / F200

The coding rules must be learned by you so that you can write the program for the manufacture.
The Coding of Informations and Instructions (Criteria)

One could build a computer which understands instructions in normal language. This would bring about quite some disadvantages:

<table>
<thead>
<tr>
<th>Language information</th>
<th>Criteria</th>
<th>Demands for coding</th>
</tr>
</thead>
<tbody>
<tr>
<td>Move the longitudinal slide - main spindle being switched on - with a given feed a distance of 25 mm at an angle of 37°.</td>
<td>1 It would be necessary to build a computer for each language (or even for each slang)</td>
<td>- Language neutral</td>
</tr>
</tbody>
</table>
|                      | 2 The long instructions are complicated and vague. | - Simple coding  
- Clear expression |
|                      | 3 The language is practice oriented. This should also be true for CNC-instructions. | - Practice-oriented |
|                      | 4 The code should be applicable to many different machine types. | - Universally applicable |

When setting up standards for the program structure of CNC-machines the aim of the many experts was to create codes for instructions which should be

- as short as possible
- simple
- language neutral
- practice-oriented
- applicable to all machines.
Program Structure

Coding of the movements
Introduction of the Cartesian Coordinates System.

Write down the instruction which you would have to give for milling. The milling spindle is on.

Number the instructions consecutively.
Coding of slide movements

The instructions

1. Move the **vertical slide** downwards (15 mm)
2. Move the **longitudinal slide** to the **left** (50 mm)
3. Move the **cross slide** forward (30 mm)

are neither short nor language-neutral nor simple.

The movements are described using the axis denomination of the Carthesian Coordinates System.

For vertical mills

X-movement: longitudinal slide
Y-movement: cross slide
Z-movement: vertically

Instruction on direction

is achieved using \( \pm \) sign.

Coded instructions

1. \(-Z\) \hspace{1cm} 15 mm
2. \(+X\) \hspace{1cm} 50 mm
3. \(-Y\) \hspace{1cm} 30 mm
The movement 1 is different to movements 2 and 3.

Movement 1

No chip removal

Speed as large as possible.

Coding:

Rapid traverse = G00

Movements 2 and 3

Straight movement and chip removal

Feed rate has to be set (depending on cutter dia., raw material, depth of cut etc.).

Coding

Linear interpolation = G01
Description of Path Lengths for Slide Movements

Also in this case simple arrangements are made. The statement "mm" (Millimeter) is left out. Only the number is written.

$X = -45.325$ means: traverse $-45.325$ mm in X-direction.

On the F1-CNC path lengths are programmed without decimal point in $1/100$ mm or $1/1000$ inch.

Thus, $23.25$ mm is programmed $2325$ and $1.253$ inch is programmed $1253$.

Sign

Measures without signs are automatically "+" measures.

The Program Sheet

All informations and instructions are entered into the program sheet. Further explanations on the following page.
The CNC-Program (structure)

The program is written down in the program manuscript.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J) (D)</th>
<th>Y (K) (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>00</td>
<td>-3.000</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>01</td>
<td>01</td>
<td>0</td>
<td>-2.500</td>
<td>0</td>
<td>120</td>
</tr>
<tr>
<td>02</td>
<td>01</td>
<td>1.050</td>
<td>0</td>
<td>0</td>
<td>120</td>
</tr>
<tr>
<td>03</td>
<td>01</td>
<td>0</td>
<td>-1.680</td>
<td>100</td>
<td>120</td>
</tr>
<tr>
<td>04</td>
<td>03</td>
<td>2.000</td>
<td>2.000</td>
<td>120</td>
<td></td>
</tr>
<tr>
<td>05</td>
<td>00</td>
<td>0</td>
<td>5.50</td>
<td>150</td>
<td></td>
</tr>
</tbody>
</table>

The program manuscript

All essential data for the manufacture of a workpiece are filled in. The composition of this program is called programming. The structure of such a program is standardized.

Parts of a program

1. The block

The program consists of blocks. A block contains all data necessary to execute an operation (i.e. order: move longitudinal slide straight on 25 mm, speed 120 mm/min).

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J) (D)</th>
<th>Y (K) (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>00</td>
<td>-3.000</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>01</td>
<td>01</td>
<td>0</td>
<td>-2.500</td>
<td>0</td>
<td>120</td>
</tr>
<tr>
<td>02</td>
<td>01</td>
<td>1.050</td>
<td>0</td>
<td>0</td>
<td>120</td>
</tr>
<tr>
<td>03</td>
<td>01</td>
<td>0</td>
<td>-1.680</td>
<td>100</td>
<td>120</td>
</tr>
</tbody>
</table>

2. The words

Each block consists of various words. Each word consists of a letter and a combination of numbers, e.g. NO1.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J) (D)</th>
<th>Y (K) (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>00</td>
<td>-3.000</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>01</td>
<td>01</td>
<td>0</td>
<td>-2.500</td>
<td>0</td>
<td>120</td>
</tr>
<tr>
<td>02</td>
<td>01</td>
<td>1.050</td>
<td>0</td>
<td>0</td>
<td>120</td>
</tr>
</tbody>
</table>

3. The word

A word consists of a letter and a combination of numbers. The letter is called address.

G 01

Address **Combination of numbers**
The Address Words of the Program Sheet/F1-CNC

1. The N-address:

\[ N = \text{abbreviation of number} \]

The instructions and informations are numbered. We talk about block number. On the F1-CNC: NOOO up to N221.

2. The G-address:

Into this column we enter the key information, i.e. the G-function or preparatory function. You will get to know the various G-functions in the course of our exercises.

3. The X,Y,Z-addresses:

They are the columns for the path data. F1-CNC:

The paths are programmed without decimal point in 1/100 mm and/or 1/1000".

4. The F-address:

F stands for "feed". For each chip removal movement the appropriate feed has to be programmed.

F1-CNC:

The feed is programmed in mm/min or 1/10 inch/min.

5. The M-address:

M stands for "miscellaneous". M-functions are called "auxiliary functions". The M-values are entered into the G-column.
6. **The D-address:**

   The cutter radius is described under D.
   Radius 5 mm → D 500 (compare M06 Tool compensation).

7. **The S-address:**

   S stands for speed.
   2000 rpm → S 2000 (compare M06).

8. **The T-address:**

   T stands for tool.
   Tool number 2 → T02 (compare tool lengths compensation).

9. **The J,K-addresses:**

   J,K are parameters for circle programming.
   These addresses are described in chapter G02/G03.

10. **The L-address:**

    is a jump address; compare G25, G27.
Standardization of Axis Systems for CNC-Machines

The axis systems are standardized for the various types of machinery according to ISO 841 and DIN 66217. The basis is the Carthesian Coordinates System (clockwise).

The right-hand rule can be of quite some help: it shows the position of the axes to one another.

Making Programming Easier

Mix-ups are quite common when programming X,Y,Z and the +/- directions. So even quite experienced programmers use auxiliary devices. Use the model of the coordinates system and you will commit less mistakes.
Axis System Milling Machines

Milling machines and machining centers are of different construction typology.

Example: Vertical mill type 1

Milling head with tool moves.

The mounted workpiece carries out longitudinal and cross movements.

Vertical mill type 2

Milling head with cutter is fixed.

The mounted workpiece carries out longitudinal, cross and vertical movements.
Description of Cutter Path

If you would have to directly describe the slide movements, it would need a continuous rethinking with the various different machine construction types.

Example: Drilling a hole

Type 1: Move milling head downwards.

Type 2: Move vertical slide upwards.

A confusing situation.

Thus, the important simple statement for CNC-machines!

The path of the cutter is described. For the programming it is all the same, whether the slides or the tool move during manufacture.
Milling programs on vertical or horizontal mills are different. The Z-axis is always the main spindle axis.

A minus Z-movement is always a feed-in movement into the workpiece (e.g. drilling).
The path information is given from a zero reference point.

Each point (place) is the reference point (place) for the following measurements.
Dimensions of Drawings

There are different types of dimensioning in technical drawings.

**Incremental dimensioning**

Starting point for the dimensioning of the next point is always the actual point which was described last.

**Absolute dimensioning**

Zero-point for the dimensioning of all points is a remaining fixed point.

**Mixed dimensioning**

In most technical drawings you find both types of dimensioning. Some measures are given from one common point (absolute) or in the incremental mode (from the actual point described last).
The Modes of Programming

It was the aim to achieve a very simple description of the traverse movements.

You can program the points and traverse movements in two different modes - so to avoid changing of dimensions in the drawing.

To instruct the computer how to calculate the values it is necessary to give a key information.

This is achieved by a G-instruction.

G90
- Absolute mode description
- Absolute mode programming (reference point programming)

- You start from one point and describe all other points.
- The zero-point of the coordinates system can be defined by you.

G91
- Incremental mode description
- Incremental mode programming

- You describe point 1 starting from point 0.
- You describe point 2 starting from point 1.
- You describe point 3 starting from point 2, etc.

You have to imagine the coordinates system shifted into the relative point.
When do you have to give the G90/G91 information to the computer?
The initial status of a CNC-machine

When you switch on the main switch the machine is in mode of operation "hand operation" = initial status.

If you press the \text{H/C} key, the mode of operation is switched to "CNC-operation".

The "initial status" of the control is incremental. All traverse movements are calculated in incremental mode.

\textbf{G90} — Absolute value programming

G90 has to be programmed.

\textbf{G91} — Incremental value programming

You may program G91, however it is not necessary since the control calculates incrementally by itself.

G90 is a self-maintaining modal function. It is valid until it is revoked, i.e. until G91 is programmed.

G91 is a self-maintaining modal function. G91 is revoked by G90.
Exercise

Describe points P1, P2, P3, P4, P5 as absolute data.

Write in block N000 the information for the mode of programming.

Exercise

Describe points P1, P2, P3, P4, P5 as incremental data.
Determining the Coordinates for Programming in Absolute Mode

Determining the workpiece zero-point in the technical drawing

In technical drawings the measures are often taken from one reference point.

For programming it is convenient that as many measures as possible can be taken over from the drawing - without calculation work.

You as programmer can determine the zero-point of the workpiece. The ideal choice can best be seen in the workpiece drawing.

Symbol

Short description
- Where to set the workpiece zero-point is your own decision.

- Pay attention to the signs of the axis.

- Write axis signs and ± signs in the drawings not described.
The origin of the coordinates system can be positioned in any point.

Points may be positioned in any of the 8 squares.

Describe the points in absolute and incremental mode.

**X – Y plane = Underneath side of workpiece**
X – Y Plane in Center of Body

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Absolute

Incremental
Informations to the Control concerning the Workpiece zero-point

You can instruct the control with G90/G91 how it should calculate the movements - in absolute or incremental mode.

Absolute value programming

Where is the origin of the coordinates system situated?

The control unit of a CNC-machine can neither see nor think.

- It does not know the position of the workpiece mounted to the slide.
- It cannot read the technical drawing and thus cannot know the position of the workpiece zero-point chosen by you.

CNC-solution:

We have to instruct the control where we want the origin of coordinates.
Fixing the Origin of the Coordinates on the F1-CNC  
(Workpiece zero-point)

Possibility 1:
Fixing with G90

If the computer receives a G90 instruction in the course of the program, it considers the actual slides position as zero-point.

In the left side mentioned situation you could not take any workpiece measures from the drawing. You would have to calculate.

This is only useful if you shift the origin of the coordinates system to the workpiece zero-point.

Example:
You move the cutter to the zero-point chosen by you. If the cutter is in this position you program G90. The origin of the coordinates is set.
Fixing the Zero-point of Coordinates with G92

G92 – Programmed offset of reference point

- We have set the workpiece zero-point.
- The cutter position is known to you (distance workpiece zero-point to cutter).

Information to computer with G92

You describe the cutter position looked at from the workpiece zero-point. In this way you fix the workpiece zero-point selected by you.

Attention:
- G92 is an information, no instruction to traverse.
- G92 means automatically absolute value programming.
- The zero-point of the workpiece can be set off with G92 within a program as often as wanted.

Format G92

\[
\begin{align*}
\text{N3/G92/X} & \pm 5/\text{Y} \pm 4/\text{Z} \pm 5 \\
(\text{vertikal}) & \\
\text{N3/G92/X} & \pm 4/\text{X} \pm 5/\text{Z} \pm 5 \\
(\text{horizontal}) & 
\end{align*}
\]
Exercises

Program the workpiece zero-point

Program the tool to the workpiece zero-point.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (O)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (O)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

4.63
Exercises

Program the workpiece zero-point

Program the indicated traverse paths.
Exercises

Program the workpiece zero-point

Program the indicated traverse paths.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (G)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Exercises

Program the workpiece zero-point

Program the indicated traverse paths.
Various Workpiece Zero-Points in one Program

By a new programming of the workpiece zero-point the previous workpiece zero-point is cancelled.

Sometimes it is easier for the programming to set various workpiece zero-points within one program.

Example:
- W1 is programmed. Plane 1 is worked on.
- Traverse cutter to starting position.
- W2 is programmed. Plane 2 is worked on.

Note:
In most cases it is best to program the reference point offset from one and the same point so that the program stays distinct.
Exercises

Program the zero-points and the paths indicated.
Mixed Programming

You may change also within one and the same program the programming mode from absolute to incremental and vice-versa.
Programming of the originally fixed workpiece zero-point

If you want to fix the originally programmed workpiece zero-point you have to either

- move the tool into the original workpiece zero-point and then program G90

or

- describe from the original workpiece zero-point the actual cutter position.
Connection:

G92 – Zero-point offset
M06 – Tool lengths compensation

M06
The Z information is an incremental target information within an independent coordinates system.

G92
With G92 you fix the origin of the coordinates system.
4. Setting up the program:
Carry out offsetting of workpiece +z-zero-point

Manufacture

1. Mounting the workpiece

We assume that you have to manufacture a few workpieces of same shape. You mount the workpiece such that it is always in the same position on the machine table.

- The machine vice is clamped.
- In Y-direction the workpiece remains always in same position because of the unmovable jaw.
- In X-direction by a stop,
- In Z-direction by identical spacers.

2. You scratch the three reference surfaces and move the tool to the program start point (= program end point, = tool change point).
Some tips for procedure

1. Determining the workpiece zero-point in the drawing:

You can see in your workpiece drawing what the best position for the workpiece zero-point will be. You determine the workpiece zero-point in your drawing.

2. Determining the starting point of the program.

3. Measuring of tools - Putting in data into a data sheet if more tools are used.

<table>
<thead>
<tr>
<th>d</th>
<th>40</th>
<th>10</th>
<th>16</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>20</td>
<td>5</td>
<td>8</td>
</tr>
<tr>
<td>F</td>
<td>80</td>
<td>160</td>
<td>40</td>
</tr>
<tr>
<td>t</td>
<td>0.7</td>
<td>5</td>
<td>8</td>
</tr>
<tr>
<td>S</td>
<td>1400</td>
<td>2000</td>
<td>2000</td>
</tr>
<tr>
<td>HZ</td>
<td>0</td>
<td>650</td>
<td>-320</td>
</tr>
</tbody>
</table>
The Miscellaneous or Switching Functions

Switching operations are programmable too on CNC-machines. The M-address is used to program them. The word for the miscellaneous functions contains a 2-digit key number.

Extract from codes for miscellaneous functions
(DIN 66025, part 2)

<table>
<thead>
<tr>
<th>Miscellaneous Function</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>Programmed stop</td>
</tr>
<tr>
<td>M01</td>
<td>Optional (planned) stop</td>
</tr>
<tr>
<td>M02</td>
<td>End of program</td>
</tr>
<tr>
<td>M03</td>
<td>Spindle clockwise</td>
</tr>
<tr>
<td>M04</td>
<td>Spindle counterclockwise</td>
</tr>
<tr>
<td>M05</td>
<td>Spindle off</td>
</tr>
<tr>
<td>M06</td>
<td>Tool change</td>
</tr>
<tr>
<td>M07</td>
<td>Coolant no. 2 ON</td>
</tr>
<tr>
<td>M08</td>
<td>Coolant no. 1 ON</td>
</tr>
<tr>
<td>M09</td>
<td>Coolant off</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Miscellaneous Function</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>M10</td>
<td>Clamp</td>
</tr>
<tr>
<td>M11</td>
<td>Unclamp</td>
</tr>
<tr>
<td>M19</td>
<td>Oriented spindle stop</td>
</tr>
<tr>
<td>M30</td>
<td>End of program</td>
</tr>
<tr>
<td>M31</td>
<td>Interlock bypass</td>
</tr>
<tr>
<td>M48</td>
<td>Constant speed on</td>
</tr>
<tr>
<td>M49</td>
<td>Constant speed off</td>
</tr>
<tr>
<td>M58</td>
<td>Workpiece change</td>
</tr>
</tbody>
</table>

All key numbers not mentioned are temporarily or permanently available. The manufacturer of the control can assign the key numbers to a given function.
M-Functions

Miscellaneous or Switching Functions on the F1-CNC

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J,D)</th>
<th>Y (K,S)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>M06</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Programming

The M key numbers are entered into the G-column.

So if there is a M-key number to be entered always add the letter M.

Input of M-values

Press M-key then put in number value.

M-Functions in standard version on F1-CNC

M00 - Programmed stop
M30 - Program end with re-set
M06 - Tool lengths compensation
      Tool data
      Tool change
M17 - Jump back instruction
M99 - Circle parameter

M-Functions with the DNC-Interface (accessory)

M03 - Spindle clockwise
M05 - Spindle counterclockwise

M08
M09
M20
M29
M22
M23

Freely available M-functions

For details compare chapter 7
Description of Block Formats

Depending on the G-functions you have to program different addresses (enter values for N,X,Y,Z,F,M,T,D,S,L,J,K into the columns).

For a better overview the single prescriptions are abbreviated.

1. You need a block number N

   This block number can be 3-digit.
   Abbreviation: N3

2. The G-address

   The G-address has two decades; it determines which addresses have to be programmed.

3. X,Y,Z-addresses

   X,Y,Z addresses may have ± signs.
   Vertical milling machine: 
   X±5, y±4, z±5
   Horizontal milling machine 
   x±4, y±5, z±4

4. F-address (feed)

   3 digits, therefore T3

5. J,K-addresses (circle parameter)

   2 digits, therefore J2, K2

6. M-address (auxiliary function)

   2 digits, therefore M2

7. T-address (tool number)

   3 digits, therefore T3

8. D-address (cutter radius)

   5 digits, therefore D5

9. S-address (speed)

   4 digits, therefore S4

10. L-address (jump)

    3 digits, therefore L3

11. H-address (with M26)

    3 digits, therefore H3

Example of a format description:
Format G00
N3/G00/X ± 5/Y ± 4/Z ± 5
Types of Controls of CNC-Machine Tools

1. Point-to-Point Control

- The tool can move only from point to point.
- The speed of the tool movement is not registered.
- The tool path from point to point is not prescribed. Only the final position has to be correct.

Application:
Drilling machines, spot welding machines
Today rather seldom in use, because most controls offer straight line or contouring characteristics at the same price.

2. Straight Line Control

The tool moves with
- given speed
- axis parallel.

During the traverse movement milling is possible.
With milling machines either
- the longitudinal slide or
- the cross slide or
- vertical slide moves, but never two slides together!

Application:
Today hardly in use anymore; replaced by contouring control.
3. Contouring Control

Various axes traverse simultaneously with a programmed feed speed on a prescribed path. The movement can be a straight line or circular movement. Nearly all CNC-machine tools are today equipped with a contouring control.

Types of Contouring Controls

a) Two-Axes Contouring Control

(2D control; 2D means two-dimensional)

Application:
Lathes, simple milling machines, erosion machines, drawing machines, punch presses, etc.)
b) Two and a half Axes contouring Control

Three times 2 axes can be moved simultaneously with programmed feed speed and this on a prescribed path.

The illustrations are there to show you what is meant by three times 2 axes.

Application:
Milling machines, machining centers, flame cutting machines, etc.
c) Three-Axes Contouring Control
(3D control)

All three axes can traverse simultaneously on a prescribed path with programmed feed speed.

Application:
Milling machines for the production of complex three-dimensional workpieces.
If you traverse in three axes simultaneously you need special milling cutters (round head cutters etc.).

Note:
There are misunderstandings caused by commonly used technical terms. A milling machine features 3 directions of movements:
- longitudinal slide movement
- cross slide movement
- vertical movement (up and down)
This is called a 3-axes machine. However, this does not imply that the machine is equipped with a 3D contouring control (3-axes contouring control).
Programming – Geometry

— The center point path of the cutter
  — influence of the cutter radius

— Trigonometry of the right triangle

— CNC conformal lettering, calculation of missing coordinates

— Transitions straight line – circular arc tangent

— Calculations of auxiliary points
  Straight line
  Circular arc tangent
Description of the cutter path

We describe the center point path of the cutter (except G72, G45-G48)

Influence of the cutter radius:

When milling contours the cutter diameter determines the programming of the cutter path.

Auxillary points:

When programming the center points of the cutter path the target points are called auxiliary points.
When manufacturing axis-parallel contours the cutter radius has to be added to or subtracted from the contour.

With non-axis parallel contours, auxiliary points have to be calculated. For this the trigonometric functions of the right triangle will do.

In quite some cases the coordinates of crossing points have to be calculated because they are not indicated in common technical drawings. Missing coordinates are calculated on the basis of trigonometric functions.
Survey
Trigonometric functions in the right triangle

**Specification:**
The right angle (90°) is characterized with the symbol $\angle$.

Both angles $\alpha$ (Alpha) and $\beta$ (Beta) are in sum 90°.
$\alpha + \beta = 90°$

**Hypotenuse:**
Opposite side of right angle.
Abbreviation: HY

**Adjacent side (AS), opposite side (OS):**
Each angle $\alpha$ and $\beta$ has a adjacent side and a opposite side.

Adjacent side = adjacent side to angle $\alpha$ or $\beta$
Opposite side = opposite side to angle $\alpha$ or $\beta$

<table>
<thead>
<tr>
<th></th>
<th>Sine = $\frac{GK}{Hy}$</th>
<th>$\sin \alpha = \frac{a}{c}$</th>
<th>$a = c \cdot \sin \alpha$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$c = \frac{a}{\sin \alpha}$</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Cosine = $\frac{AK}{Hy}$</th>
<th>$\cos \alpha = \frac{b}{c}$</th>
<th>$b = c \cdot \cos \alpha$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$c = \frac{b}{\cos \alpha}$</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Tangent = $\frac{GK}{AK}$</th>
<th>$\tan \alpha = \frac{a}{b}$</th>
<th>$a = b \cdot \tan \alpha$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$b = \frac{a}{\tan \alpha}$</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Cotangent = $\frac{AK}{GK}$</th>
<th>$\cot \alpha = \frac{b}{a}$</th>
<th>$b = a \cdot \cot \alpha$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$a = \frac{b}{\cot \alpha}$</td>
<td></td>
</tr>
</tbody>
</table>
CNC-Conformal Lettering
The Calculation of Coordinates

In many cases the lettering of technical drawings is such that the coordinates for the CNC-programming have to be calculated.

Missing coordinates data can mostly be calculated using simple trigonometric functions.
Calculation of Coordinates

Transitions: Axis-parallel straight line — straight line at angle

The Y-coordinate of point $P_3$ is not known.

\[
tg \alpha = \frac{Y(P_2P_3)}{20}
\]

\[
Y(P_2P_3) = tg \alpha \cdot X(P_2P_3); \alpha = 30^\circ
\]

\[
= tg 30^\circ \cdot 20 = 11,54 \text{ mm}
\]

Exercise:

Calculate the missing coordinate of point $P_3$.

Make a CNC-conformal drawing.
Calculation of Coordinates

Transition straight line – tangential arc

1. Calculate the X-coordinate of S (crossing point between straight line and slant plane)

\[ \tan \alpha = \frac{X}{30} \]

\[ X = \tan 30 \times 30 = 17.32 \]

2. Calculate the X-coordinate of \( P_2 \).
3. Calculate the X- and Y-coordinate of point P₃.

\[ \overline{SP}_3 = 11.55 \text{ mm} \]

\[ \sin \alpha = \frac{X}{11.55} \]

\[ X = \sin 30^\circ \times 11.55 = 5.78 \text{ mm} \]

\[ \cos \alpha = \frac{Y}{11.55} \]

\[ Y = \cos 30^\circ \times 11.55 = 10 \text{ mm} \]

Letter all points in absolute and incremental mode
Calculation of auxiliary points

Example 1

You program the path of the milling axis Q₀/Q₁/Q₂/Q₃ ...

Points Q₁ and Q₂ have to be calculated.

Cutter dia. 10 mm.

1. Calculate the Y-coordinate of point P₂.

\[ \tan 30° = \frac{YP₂}{30} \]

\[ YP₂ = 30 \cdot \tan 30° = 17.32 \text{ mm} \]
Example 1 (continued)

2. The path from \( Q_0 \) to \( Q_1 \) is composed of:

\[ r + 20 \text{ mm} + \Delta X_1 \]

\[ \Delta X_1 = \tan \frac{\alpha}{2} \cdot r = \tan 15.5 \approx 1.34 \text{ mm} \]

\[ Q_0 Q_1 = 26.34 \text{ mm} \]

Coordinates: \( Q_0 = \text{Workpiece zero-point} \)

<table>
<thead>
<tr>
<th>( Q_0 )</th>
<th>( X )</th>
<th>( Y )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( Q_0 )</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>( Q_1 )</td>
<td>26.34</td>
<td>0</td>
</tr>
<tr>
<td>( Q_2 )</td>
<td>60</td>
<td>0</td>
</tr>
</tbody>
</table>
3. Calculation of $Y_{Q2}$

$Y_{Q2} = 17.32 - \Delta Y_2$

$tg \frac{\alpha_2}{2} = \frac{\Delta Y_2}{r}$

$\Delta Y_2 = r \cdot tg \frac{\alpha_2}{2} = 5 \cdot tg 30$

$= 2.87 \text{ mm}$

$Y_{Q2} = 17.32 - 2.87$

$= 14.45 \text{ mm}$

Dimension the auxiliary points in absolute and incremental mode. Fix the workpiece zero-point by yourself.
Exercise 1 (Calculation of auxiliary Points)

Calculate the $\Delta X$ and $\Delta Y$ values.
Exercise 1 (continued)

Dimension the auxiliary points in absolute mode. Workpiece zero-point as in drawing.
Exercise 1 (continued)

Dimension the auxiliary points in incremental mode.
Exercise 2

- Calculate the coordinate of point \( P_3 \).

- Calculate the missing auxiliary coordinates.

Cutter radius 10 mm

- Pay attention: angle \( \alpha_2 \) is given as interior angle (enclosed angle).
Exercise 3

Program the exercise in absolute or incremental mode.

Fix the workpiece zero-point and the cutter radius yourself.
**Example 2**

**Approach at angle**

(A big safety distance was selected intentionally!)

\[ \alpha_1 = 30^\circ \quad \alpha_2 = 60^\circ \]

- \( S_1 \) = Safety distance (10 mm)
- \( r \) = Cutter radius (5 mm)

**Calculation of point \( Q_1 \)**

1. \( X_1 \):
   \[ \tan \alpha_1 = \frac{S}{X_1} \]
   \[ X_1 = \frac{S}{\tan \alpha_1} = \frac{10}{\tan 30^\circ} = 17.32 \text{ mm} \]

2. \( \Delta X_1 \):
   \[ \tan \frac{\alpha_1}{2} = \frac{\Delta X_1}{r} \]
   \[ \Delta X_1 = \tan \frac{\alpha_1}{2} \cdot r = \tan 15.5^\circ \cdot 5 \approx 1.34 \text{ mm} \]

3. Distance \( y(P_1Q_1) = S \cdot r = 15 \text{ mm} \)
Example 2 (continued)

Calculation of point $Q_2$

$S_2 = 20$ mm
$r = 5$ mm
$\alpha_2 = 60^\circ$

1. $Y_2$

$\tan \alpha_2 = \frac{S}{Y_2}$

$Y_2 = \frac{S}{\tan \alpha_2} = \frac{20}{\tan 60^\circ} = 11.55$ mm

2. $\Delta Y_2$

$\tan \frac{\alpha_2}{2} = \frac{\Delta Y_2}{r}$

$\Delta Y_2 = \tan \frac{\alpha_2}{2} \cdot r = 2.89$ mm

Describe the coordinates from points $Q_1, Q_2$ in connection with $P_1, P_2$. 

Auxiliary Points with acute Angles

With acute angles you have to traverse long no-load paths from target point A to start point B.

That takes time. It may happen that the slide movements are too short or there is a collision with a chucking device or you mill into a workpiece part.

Two "short cuts" are common in milling techniques

Traverse with various straight lines.

Traverse with circular arc.
Traverse in circular arc

\[ \sin \alpha_2 = \frac{\Delta X_2}{r} \]
\[ \cos \alpha_2 = \frac{\Delta Y_2}{r} \]
\[ \Delta X_2 = \sin \alpha_2 \cdot r \]
\[ \Delta Y_2 = \cos \alpha_2 \cdot r \]
 Traverse in circular arc

Exercise:
Dimension auxiliary points absolute an incremental.
Program absolute and incremental.
Select workpiece zero-point.
Program absolute and incremental.
Select workpiece zero-point.

\[ P_1 P_2 = 40 \text{ mm} \]
\[ \alpha_2 = 30^\circ \]
Calculation of Auxiliary Points

Straight line movement
Traverse with various straight lines

Exercise:

- Dimension absolute and incremental.
- Program the paths.

\[ P_1 P_2 = 40 \text{ mm} \]
\[ \alpha_2 = 30^\circ \]
Chapter 5

Programming

The contents are arranged according to the numbering of the G-functions

G90/G91/G92    Compare chapter 4
G65/G66    Compare tape operation
RS-232 C operation
Chapter 10
Hints for the Beginner

- Program start point
  Program target point
  Tool change point

- Potting the cutter path
The Start Point of the Program

The Tool Change Point

The End Point of the Program

Just imagine the sequence of operation: the workpiece has to be mounted and dismounted; tools will have to be changed.

The start point of the program should be chosen so that all handling can be done without any obstacle.

Determination of Coordinates

Scratch or touch the reference surfaces slightly and move the tool by hand to the selected starting point.

Start Point for Chip Removal

Position the tool in a safety distance to the workpiece. So you can find out during a program run whether the tool runs into the workpiece because of a programming fault (with rapid traverse).
Auxiliary Drawings for Programming

As with the programming of turned pieces also with the programming of milled pieces the technical drawing is a valuable help. This is particularly true in the beginning. It is easier to set up and check the program.

Turned pieces:

You draw and program the path of the edge tip of the tool bit. The edge tip is the part of the tool bit which produces the contour.

The tool bit movement is in one plane, thus it is easier to depict.

Milled pieces:

Here you have to think and to draw in three dimensions. This needs quite some experience.

A three-dimensional depiction is very distinct but not easy to do. Besides that, all paths which are not parallel to axis show shortened.
A separate drawing is a great help for the first exercises.

**An example:**

1. Enter into a sketch the program start point of the cutter.

2. If you firstly move in Z-direction to the milling plane you can draw in the workpiece and the cutter path.

2.1. Mark the raw stock contour and the finished part contour.
2.2. Draw in the cutter paths. Mark the various auxiliary points. 

Draw in the direction of movement.

2.3. Number the various blocks. The checking of the program will be much easier.

3. Blocks with no traverse movements programmed can be assigned to the auxiliary points.

4. With absolute programming draw in zero-point of workpiece.
G00 – Rapid Traverse
Straight line approach movement

The target point is described from the starting point of the cutter.

The target point is described from the previously fixed zero-point of the coordinates system.
G00 - Rapid Traverse

- All movements are carried out with the highest possible speed, i.e. rapid traverse (with the F1-CNC: 600 mm/min).

- G00 is no chip removal movement but a movement without milling cutter being in action.

Programming Exercises

In order to move the milling cutter to its working position you have various possibilities.

1. Traverse only in 1 axis
The two other axes are zero. - You have six possibilities. Program all of them, absolute and incremental.
a) Incremental Value Programming:
- The milling cutter is in the position which is indicated in the drawing.
- It is moved to milling position with GOO.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

b) Absolute Value Programming:
- Milling cutter is moved to milling position.
- Program the traverse paths

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
2. Traverse in one block simultaneously in 2 axes

Program absolute and incremental. - The zero-point of the coordinate system for the absolute programming is in point $P_0$.

Draw in the possibilities.

**Question:**

How many possibilities are given if you move all three axes simultaneously?
G01 – Straight Line Interpolation

Straight line cutting movement, feed programming necessary.

Incremental programming

\[ \Delta X 25 \text{ mm} \quad \Delta Z 18 \text{ mm} \]

Absoute programming

\[ X 40 \text{ mm} \quad Z 5 \text{ mm} \quad Y 32 \text{ mm} \]

G01/X2500/Y1800/Z = 0/F ...

G01/X4000/Y3200/Z -500/F...

The target point is described from the starting point of the cutter.

The target point is described from the previously fixed zero-point of the coordinates system.
G01 – Linear Interpolation

Linear means straight lined. Interpolation means the finding of intermediate values.
- G01 is a chip removal movement.
- With each chip removal movement you have to program a feed.

**Format G01**

N3/G01/X ± 5/Y ± 4/Z ± 5/F3

With G01 you can traverse parallel to axis and at each angle in one plane.
Examples G01 (1)
Milling of a Shoulder

- Milling cutter dia. 10 mm
- Mode of programming: incremental
- A shoulder with a width of 5 mm and a depth of 4 mm has to be milled.

1. Determining the starting point as indicated.

2. Programming with G00 to the starting point of chip removal. Choose a safety distance of 5 mm.
Example (1) (continued)

Determination of the Path for the Milling Cutter

With a diameter of the milling cutter of 10 mm and a width of the shoulder of 5 mm, the axis of the cutter is exactly at the edge of the workpiece.

Programming:

Program end position is starting position.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J) (D)</th>
<th>Y (K) (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>00</td>
<td>2000</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>01</td>
<td>00</td>
<td>0</td>
<td>0</td>
<td>-3400</td>
<td></td>
</tr>
<tr>
<td>02</td>
<td>01</td>
<td>6000</td>
<td>0</td>
<td>0</td>
<td>200</td>
</tr>
<tr>
<td>03</td>
<td>01</td>
<td>0</td>
<td>5000</td>
<td>0</td>
<td>200</td>
</tr>
<tr>
<td>04</td>
<td>01</td>
<td>-5000</td>
<td>0</td>
<td>0</td>
<td>200</td>
</tr>
<tr>
<td>05</td>
<td>01</td>
<td>0</td>
<td>-5000</td>
<td>0</td>
<td>200</td>
</tr>
<tr>
<td>06</td>
<td>00</td>
<td>-3000</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>07</td>
<td>00</td>
<td>0</td>
<td>0</td>
<td>3400</td>
<td></td>
</tr>
<tr>
<td>08</td>
<td>M30</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Exercise 2 for Example 1

- Program this example in absolute values.

- Carry out a zero-point offset with G92.

- Starting position and zero-point of workpiece as in drawing.
G01 – Example 2
Milling a Groove

- Mode of programming: incremental
- Dia. of milling cutter: 10 mm
- Starting position as in drawing
- Depth of groove: 4 mm
- Feed (compare technological data)
- Safety distance before cutting: 5 mm

Pay attention:
When feeding in the cutter, halve the feed values.
**Exercise 1 for Example 2**

Write the program according to the traverse paths as indicated.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

**Exercise 2 for Example 2**

Program the example absolute with zero-point offset.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

**Exercise 3 for Example 2**

Choose other traverse paths for G00.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
G01 – Example 3
Milling a Pocket

- Milling cutter dia. 10 mm.
- Starting position as in drawing
- Safety distance before cutting 5 mm

Choose the path of the milling cutter such that there is always an overlap of 1-2 mm (in industry approx. 1/10 of the dia. of the cutter is chosen).
Drawing the Path of the Milling Cutter

Dimensioning

An important support for your programming work is an appropriate drawing.

- Enter the block number
- Mark begin and end of the block
- Use the largest possible scale when drawing.
- Dimension auxiliary measurements

Program this groove as in the drawing in absolute and incremental mode.

Programming sketch and dimensioning of auxiliary measurements for absolute programming.
Programming sketch and dimensioning of auxiliary measurements for incremental programming.
Example 4

The milling path in example 3 would leave the corners in the pocket unfinished.

With pocket milling you cut a rough pocket first. With a final cut you mill the complete contour once again to reach a better surface quality.

Exercise:
- Program and mill the given pocket.
- As final run a continuous smooth cut of 2 mm shall be taken off. Mode of programming as you wish.
- Select the zero point of the workpiece yourself.
Example 5/G01

Milling a Cross Slot of 45°

Diameter of milling cutter 8 mm.
Program the zero point of the workpiece using absolute value programming.

Make a drawing and use reference dimensions!

1. Start position: Milling
5 mm away from theoretical X-edge
5 mm away from theoretical Y-edge

2. Target position:
As indicated (X 5 mm, Z 5 mm)
Example 6: Bores $4 \times 90^\circ$

+ The center point coordinates of the bolt circle are known.

+ The coordinates of the bores have to be calculated.

\[
\sin \alpha = \frac{Y_1}{R} \\
Y_1 = R \cdot \sin 45^\circ = 15.0,707 = 10,6 \\
\cos \alpha = \frac{X_1}{R} \\
X_1 = R \cdot \cos 45^\circ = 15.0,707 = 10,6
\]

Since the bores are positioned symmetrically to the center point, you can calculate the $X,Y$ coordinates of the other bores (by adding or subtracting).

Dimension the drawing for CNC-manufacture – in absolute and incremental mode.

Program the example.
Example 7:
Bores 6 x 60°

- Calculate the coordinates of the bores.
- Dimension the part for CNC programming.
- Program example.

Incremental programming

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (T)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Example 7:
Absolute programming and lettering
Example 8: Hexagon

Use cutter dia. 16 mm

1. You calculated the coordinates of the corner points in one of the previous examples.
   Transfer the values for points 1 to 6.

2. You have to calculate the auxiliary coordinates of the cutter center path.
Example 8:
Hexagon

You have to add respectively subtract the $\Delta X$ and radius values to the coordinate values of points 1, 2, 3, 4, 5, 6.

Calculation of $\Delta X$

\[
\tan \frac{\alpha}{2} = \frac{r}{\Delta X}
\]

\[
\Delta X = \frac{r}{\tan \frac{\alpha}{2}}
\]

Put in measurements for auxiliary points. Program the example!

Pay attention whether there is remaining material at the outer corners. If yes, mill it off.
On conventional machine tools circular arcs can be produced only using special auxiliary devices. On CNC-machines circular arcs of any angle or radius can be reached without such special devices. The key information for circular arcs is G02 and G03.
G02 – Circular Interpolation Clockwise

G03 – Circular Interpolation Counterclockwise

In order to formulate what you mean by clockwise and counterclockwise, we have to determine the direction from which we look at.

**Determination**

You have always to look at the sense of rotation in one plane from the positive direction of the third axis.

**Interpolation Clockwise G02**

**XY-Plane:**
Look from +Z direction to -Z direction.

**YZ-Plane:**
Look from +X to -X.
Interpolation G02.
Clockwise

XZ-Plane:
Look from +Y to -Y.

In this technical sketch the direction in the XZ-plane seems to be inverted.
Arrows on the F1-CNC Milling Machine

Metric
Size of radii 0,01 - 99,99 mm in steps of 0,01 mm

Inch
Size of radii 0,001 - 3.999 Inch in steps of 1/1000 inch

Programming
On the F1-CNC you can program quarter arcs (90°) or arcs of circles in steps of 1°.

Programming of arcs 90° on the F1-CNC
1. The sense of rotation is described with G02/G03.
2. The end point of the quarter arc is determined by the X,Y,Z addresses - either starting from point PA (incremental) or from the workpiece zero-point (absolute).
3. The F-address is used to describe the feed.

Format
N3 \{ G02 \[ \pm 5 (\pm 4) / Y \pm 4 (\pm 5) / Z \pm 5 / F3 \]

\pm 4 resp. \pm 5 with X,Y-values for vertical resp. horizontal axis system.
Programming of Quarter Arcs in the XY-Plane

Format G02/G03

N3 G02 | X±5/Y±4/Z=0 /F3

G02 incremental Programming

Example: radius 10 mm

Programmed are X,Y values looked at from the starting point.

<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td>02</td>
<td>+1000</td>
<td>-1000</td>
<td>0</td>
<td></td>
<td>Arc 1</td>
</tr>
<tr>
<td>02</td>
<td>-1000</td>
<td>-1000</td>
<td>0</td>
<td></td>
<td>Arc 2</td>
</tr>
<tr>
<td>02</td>
<td>-1000</td>
<td>+1000</td>
<td>0</td>
<td></td>
<td>Arc 3</td>
</tr>
<tr>
<td>02</td>
<td>+1000</td>
<td>+1000</td>
<td>0</td>
<td></td>
<td>Arc 4</td>
</tr>
</tbody>
</table>

Attention:

In the XY-plane the Z-value has to be programmed with zero.
G02 – Absolute Programming

Zero-point of workpiece as indicated in drawing.

You program the XY-coordinates of the end point of quarter arc, looked at from the previously fixed point (W).

Note:

Arcs can be moved only in one plane. Thus, the Z-value of the previous block has to be taken over.

Block N01/N02: Move to start position
Block N7: Infeed in Z -100
Block N8/N9: Arcs 1,2 set deeper

<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td>000</td>
<td>92</td>
<td>0</td>
<td>0</td>
<td>1000</td>
<td></td>
</tr>
<tr>
<td>01</td>
<td>00</td>
<td>2000</td>
<td>2000</td>
<td>1000</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>01</td>
<td>2000</td>
<td>2000</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>02</td>
<td>3000</td>
<td>1000</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>02</td>
<td>2000</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>02</td>
<td>1000</td>
<td>1000</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>02</td>
<td>2000</td>
<td>2000</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>01</td>
<td>2000</td>
<td>2000</td>
<td>-100</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>02</td>
<td>3000</td>
<td>1000</td>
<td>-100</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>02</td>
<td>2000</td>
<td>0</td>
<td>-100</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Position milling cutter at start G02

Format G02/G03

N03 G02 X ± 5/Y ± 4/Z ± 5/F3
**Exercises**

**G03 - Incremental Programming**
- Position of milling cutter at start as indicated in drawing.
- Circle is in XY-plane Z=0
- Start the circle programming in point "0".

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**G03 - Absolute Programming**
- Position of milling cutter at start as indicated in drawing.
- Carry out offset of zero point.
- Circle is parallel in XY-plane, but at a distance Z +10 mm.
- Start the circle programming in point "0".

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Programming Exercise G02/G03

- Mode of Programming: incremental
- Approach direction as in drawing
- Determine starting point yourself
- Determine drawing with dimensioning of triangulation (station).

Approach direction as in drawing.
Programming Exercise G02/G03

Alternative 1

- Mode of Programming: absolute
- Zero-point of workpiece as in drawing.
- Starting point of milling cutter as in drawing.
- Dia. of milling cutter 10 mm.

Alternative 2

- Mode of programming: absolute
- Zero-point of workpiece as in drawing
- Starting point as in drawing.
Exercise
Mode of programming: incremental

- Circle in YZ-plane
- Start point as in drawing

Exercise
Mode of programming: absolute

- Zero-point as in drawing
- Start point and end point for programming is workpiece zero-point.
Circles X-Z Plane

Exercise

- Mode of programming: incremental
- Starting point as in drawing

<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Exercise

- Mode of programming: absolute
- Zero-point as in drawing
- Starting point and end point for programming is the zero-point.

<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Some Terms for Circular Interpolation G02/03

Complete circle programming
A circle up to 360° can be programmed in one block.

Quadrants programming
A circle is divided into 4 quadrants. In one block only one arc of max. 90° can be programmed. The arc of circle has to be within a given quadrant.

In this case two blocks are necessary because the arc reaches over 2 quadrants.

Fl-CNC
- Quadrants programming
- To program a part of an arc within a quadrant, a code in two blocks is used.
Arcs with Angles at Random

On the F1-CNC arcs in steps of 1° each can be programmed. The programming is done in various subsequent blocks.

Mode of programming: incremental
(The following examples are in the XY-plane; for all other planes this principle is valid too).
Radius 10 mm

First block
Here the 90° arc in which the partial arc circle is situated will be determined.

N100/G02/X1000/Y -1000/Z .../F ...

With G02 the computer is given information on the sense of rotation.

With X 1000/Y -1000 the computer knows the quadrant (sign of X,Y) and the radius of the arc.

Next block
N101/M99/J = 0/K = 30

M99 is the key information for the arc ≠ 90°.

J-address: for the grades statement of the start of the arc within the quadrant.

K-address: target address of the arc.
Statement in grades.

Blocks N100/101 are considered by the computer to be one unit. The computer asks whether there is a M99 instruction in the block following a G02/G03 instruction.
Example
Incremental value programming

N100/G02/X1000/Y -1000/Z=0/F...
N101/M99/J24/K67

Arc of circle reaching over a few quadrants.

N102/G02/X1000/Y -1000/Z=0/F...
Arc in quadrant II.

N103/G02/X -1000/Y -1000/Z=0/F...
Arc in quadrant III.
Using the Chart

The chart shows you the J,K-values, the exact grades and the coordinates of points for a circle with radius 1.

In order to program the cutter path it is often necessary to calculate the coordinates of the arc starting (PA) and target point (P2). These points are missing in many drawings.

(All examples are in the X,Y-plane, the same principle is valid for all other planes too)

Example:

X(a) and Y(b) coordinates of the target point (P2) are not known.

Calculation: a

\[ a = R \cdot \cos 46.01 = \frac{1}{R} \]

\[ l = R \cdot \cos 46.01 = 6.9453 \]

\[ a = 10 - 6.945 = 3.0567 \]

Calculation: b

\[ \sin 46.01 = \frac{b}{R} \]

\[ b = R \cdot \sin 46.01 = 7.194 \]

These values can also be read from the chart.
## Circular Interpolation – Parameter

### XYZ-Values at the Circle 1

<table>
<thead>
<tr>
<th>J,K</th>
<th>Grad 0</th>
<th>XYZ 0</th>
<th>XYZ 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.03</td>
<td>0</td>
<td>191</td>
</tr>
<tr>
<td>2</td>
<td>1.98</td>
<td>0</td>
<td>347</td>
</tr>
<tr>
<td>3</td>
<td>3.02</td>
<td>14</td>
<td>528</td>
</tr>
<tr>
<td>4</td>
<td>4.06</td>
<td>26</td>
<td>708</td>
</tr>
<tr>
<td>5</td>
<td>5.10</td>
<td>42</td>
<td>989</td>
</tr>
<tr>
<td>6</td>
<td>6.05</td>
<td>56</td>
<td>1056</td>
</tr>
<tr>
<td>7</td>
<td>7.01</td>
<td>69</td>
<td>1222</td>
</tr>
<tr>
<td>8</td>
<td>8.06</td>
<td>97</td>
<td>1403</td>
</tr>
<tr>
<td>9</td>
<td>9.03</td>
<td>125</td>
<td>1569</td>
</tr>
<tr>
<td>10</td>
<td>9.39</td>
<td>153</td>
<td>1736</td>
</tr>
<tr>
<td>11</td>
<td>10.96</td>
<td>181</td>
<td>1903</td>
</tr>
<tr>
<td>12</td>
<td>11.33</td>
<td>203</td>
<td>2069</td>
</tr>
<tr>
<td>13</td>
<td>12.39</td>
<td>250</td>
<td>2259</td>
</tr>
<tr>
<td>14</td>
<td>14.05</td>
<td>292</td>
<td>2431</td>
</tr>
<tr>
<td>15</td>
<td>15.03</td>
<td>333</td>
<td>2597</td>
</tr>
<tr>
<td>16</td>
<td>16.02</td>
<td>375</td>
<td>2764</td>
</tr>
<tr>
<td>17</td>
<td>17.02</td>
<td>431</td>
<td>2931</td>
</tr>
<tr>
<td>18</td>
<td>18.03</td>
<td>486</td>
<td>3097</td>
</tr>
<tr>
<td>19</td>
<td>19.03</td>
<td>542</td>
<td>3264</td>
</tr>
<tr>
<td>20</td>
<td>20.04</td>
<td>597</td>
<td>3431</td>
</tr>
<tr>
<td>21</td>
<td>20.37</td>
<td>653</td>
<td>3583</td>
</tr>
<tr>
<td>22</td>
<td>22.00</td>
<td>722</td>
<td>3750</td>
</tr>
<tr>
<td>23</td>
<td>23.04</td>
<td>792</td>
<td>3917</td>
</tr>
<tr>
<td>24</td>
<td>24.00</td>
<td>861</td>
<td>4069</td>
</tr>
<tr>
<td>25</td>
<td>24.96</td>
<td>931</td>
<td>4222</td>
</tr>
<tr>
<td>26</td>
<td>25.32</td>
<td>1000</td>
<td>4375</td>
</tr>
<tr>
<td>27</td>
<td>26.39</td>
<td>1083</td>
<td>4542</td>
</tr>
<tr>
<td>28</td>
<td>27.38</td>
<td>1167</td>
<td>4684</td>
</tr>
<tr>
<td>29</td>
<td>28.98</td>
<td>1250</td>
<td>4847</td>
</tr>
<tr>
<td>30</td>
<td>29.98</td>
<td>1333</td>
<td>5000</td>
</tr>
<tr>
<td>31</td>
<td>30.97</td>
<td>1417</td>
<td>5153</td>
</tr>
<tr>
<td>32</td>
<td>32.01</td>
<td>1514</td>
<td>5306</td>
</tr>
<tr>
<td>33</td>
<td>33.05</td>
<td>1611</td>
<td>5458</td>
</tr>
<tr>
<td>34</td>
<td>34.02</td>
<td>1708</td>
<td>5597</td>
</tr>
<tr>
<td>35</td>
<td>34.99</td>
<td>1806</td>
<td>5736</td>
</tr>
<tr>
<td>36</td>
<td>35.96</td>
<td>1903</td>
<td>5875</td>
</tr>
<tr>
<td>37</td>
<td>36.93</td>
<td>2000</td>
<td>6014</td>
</tr>
<tr>
<td>38</td>
<td>37.95</td>
<td>2111</td>
<td>6153</td>
</tr>
<tr>
<td>39</td>
<td>38.97</td>
<td>2222</td>
<td>6292</td>
</tr>
<tr>
<td>40</td>
<td>39.98</td>
<td>2333</td>
<td>6431</td>
</tr>
<tr>
<td>41</td>
<td>41.00</td>
<td>2444</td>
<td>6569</td>
</tr>
<tr>
<td>42</td>
<td>41.96</td>
<td>2556</td>
<td>6694</td>
</tr>
<tr>
<td>43</td>
<td>42.97</td>
<td>2681</td>
<td>6819</td>
</tr>
<tr>
<td>44</td>
<td>43.86</td>
<td>2806</td>
<td>6944</td>
</tr>
<tr>
<td>45</td>
<td>45.00</td>
<td>2931</td>
<td>7053</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>J,K</th>
<th>Grad 0</th>
<th>XYZ 0</th>
<th>XYZ 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>46</td>
<td>46.01</td>
<td>3056</td>
<td>7194</td>
</tr>
<tr>
<td>47</td>
<td>47.02</td>
<td>3181</td>
<td>7319</td>
</tr>
<tr>
<td>48</td>
<td>48.03</td>
<td>3306</td>
<td>7444</td>
</tr>
<tr>
<td>49</td>
<td>48.99</td>
<td>3431</td>
<td>7556</td>
</tr>
<tr>
<td>50</td>
<td>50.01</td>
<td>3569</td>
<td>7677</td>
</tr>
<tr>
<td>51</td>
<td>51.02</td>
<td>3708</td>
<td>7778</td>
</tr>
<tr>
<td>52</td>
<td>52.04</td>
<td>3847</td>
<td>7869</td>
</tr>
<tr>
<td>53</td>
<td>53.06</td>
<td>3986</td>
<td>8080</td>
</tr>
<tr>
<td>54</td>
<td>54.03</td>
<td>4125</td>
<td>8097</td>
</tr>
<tr>
<td>55</td>
<td>55.00</td>
<td>4264</td>
<td>8194</td>
</tr>
<tr>
<td>56</td>
<td>55.97</td>
<td>4403</td>
<td>8292</td>
</tr>
<tr>
<td>57</td>
<td>56.94</td>
<td>4542</td>
<td>8389</td>
</tr>
<tr>
<td>58</td>
<td>57.98</td>
<td>4634</td>
<td>8486</td>
</tr>
<tr>
<td>59</td>
<td>59.02</td>
<td>4847</td>
<td>8583</td>
</tr>
<tr>
<td>60</td>
<td>60.01</td>
<td>5000</td>
<td>8657</td>
</tr>
<tr>
<td>61</td>
<td>61.01</td>
<td>5153</td>
<td>8750</td>
</tr>
<tr>
<td>62</td>
<td>62.01</td>
<td>5306</td>
<td>8833</td>
</tr>
<tr>
<td>63</td>
<td>63.00</td>
<td>5458</td>
<td>9017</td>
</tr>
<tr>
<td>64</td>
<td>64.07</td>
<td>5625</td>
<td>9000</td>
</tr>
<tr>
<td>65</td>
<td>65.03</td>
<td>5773</td>
<td>9063</td>
</tr>
<tr>
<td>66</td>
<td>65.99</td>
<td>5931</td>
<td>9139</td>
</tr>
<tr>
<td>67</td>
<td>66.35</td>
<td>6083</td>
<td>9208</td>
</tr>
<tr>
<td>68</td>
<td>67.99</td>
<td>6250</td>
<td>9278</td>
</tr>
<tr>
<td>69</td>
<td>69.02</td>
<td>6417</td>
<td>9347</td>
</tr>
<tr>
<td>70</td>
<td>69.95</td>
<td>6569</td>
<td>9403</td>
</tr>
<tr>
<td>71</td>
<td>70.96</td>
<td>6736</td>
<td>9458</td>
</tr>
<tr>
<td>72</td>
<td>71.96</td>
<td>6903</td>
<td>9514</td>
</tr>
<tr>
<td>73</td>
<td>72.97</td>
<td>7069</td>
<td>9563</td>
</tr>
<tr>
<td>74</td>
<td>73.97</td>
<td>7236</td>
<td>9625</td>
</tr>
<tr>
<td>75</td>
<td>74.96</td>
<td>7403</td>
<td>9667</td>
</tr>
<tr>
<td>76</td>
<td>75.94</td>
<td>7569</td>
<td>9708</td>
</tr>
<tr>
<td>77</td>
<td>77.00</td>
<td>7758</td>
<td>9750</td>
</tr>
<tr>
<td>78</td>
<td>78.01</td>
<td>7931</td>
<td>9792</td>
</tr>
<tr>
<td>79</td>
<td>79.03</td>
<td>8097</td>
<td>9819</td>
</tr>
<tr>
<td>80</td>
<td>80.00</td>
<td>8264</td>
<td>9847</td>
</tr>
<tr>
<td>81</td>
<td>80.96</td>
<td>8431</td>
<td>9875</td>
</tr>
<tr>
<td>82</td>
<td>81.93</td>
<td>8597</td>
<td>9931</td>
</tr>
<tr>
<td>83</td>
<td>82.96</td>
<td>8772</td>
<td>9991</td>
</tr>
<tr>
<td>84</td>
<td>83.94</td>
<td>8944</td>
<td>9994</td>
</tr>
<tr>
<td>85</td>
<td>84.93</td>
<td>9111</td>
<td>9958</td>
</tr>
<tr>
<td>86</td>
<td>85.93</td>
<td>9292</td>
<td>9972</td>
</tr>
<tr>
<td>87</td>
<td>86.97</td>
<td>9472</td>
<td>9986</td>
</tr>
<tr>
<td>88</td>
<td>88.01</td>
<td>9653</td>
<td>10000</td>
</tr>
<tr>
<td>89</td>
<td>89.96</td>
<td>9819</td>
<td>10000</td>
</tr>
<tr>
<td>90</td>
<td>90.00</td>
<td>10000</td>
<td>10000</td>
</tr>
</tbody>
</table>
In the charts the a, b values are indicated for the standard circle in 4 digits.

### Example

**Radius 1 mm**

25° (24.96°)

- **a-value**: 0.0931 mm
- **b-value**: 0.4222 mm

### Values (a, b) for any desired angle (random)

Multiply a, b values with radius sizes.

**Example**

\( \alpha = 41° \)

Radius 6.35 mm

- a = 0.2444 x 6.35 = 1.55194
- b = 0.6569 x 6.35 = 4.171315

The values must be programmed without rounding off.

- a \( \uparrow 155 \)
- b \( \uparrow 417 \)

<table>
<thead>
<tr>
<th>Angle</th>
<th>a-value</th>
<th>b-value</th>
</tr>
</thead>
<tbody>
<tr>
<td>16°</td>
<td>16.02</td>
<td>1417</td>
</tr>
<tr>
<td>17°</td>
<td>17.02</td>
<td>1417</td>
</tr>
<tr>
<td>18°</td>
<td>18.00</td>
<td>1514</td>
</tr>
<tr>
<td>19°</td>
<td>19.01</td>
<td>1514</td>
</tr>
<tr>
<td>20°</td>
<td>20.04</td>
<td>1514</td>
</tr>
<tr>
<td>21°</td>
<td>20.27</td>
<td>1514</td>
</tr>
<tr>
<td>22°</td>
<td>22.00</td>
<td>1514</td>
</tr>
<tr>
<td>23°</td>
<td>23.04</td>
<td>1514</td>
</tr>
<tr>
<td>24°</td>
<td>24.00</td>
<td>1514</td>
</tr>
<tr>
<td>25°</td>
<td>24.96</td>
<td>1514</td>
</tr>
<tr>
<td>26°</td>
<td>25.92</td>
<td>1514</td>
</tr>
<tr>
<td>27°</td>
<td>26.99</td>
<td>1514</td>
</tr>
<tr>
<td>28°</td>
<td>27.98</td>
<td>1514</td>
</tr>
<tr>
<td>29°</td>
<td>28.98</td>
<td>1514</td>
</tr>
<tr>
<td>30°</td>
<td>29.98</td>
<td>1514</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Angle</th>
<th>a-value</th>
<th>b-value</th>
</tr>
</thead>
<tbody>
<tr>
<td>31°</td>
<td>30.37</td>
<td>1417</td>
</tr>
<tr>
<td>32°</td>
<td>32.01</td>
<td>1514</td>
</tr>
<tr>
<td>33°</td>
<td>33.05</td>
<td>1514</td>
</tr>
<tr>
<td>34°</td>
<td>34.02</td>
<td>1514</td>
</tr>
<tr>
<td>35°</td>
<td>34.99</td>
<td>1514</td>
</tr>
<tr>
<td>36°</td>
<td>35.96</td>
<td>1514</td>
</tr>
<tr>
<td>37°</td>
<td>36.93</td>
<td>1514</td>
</tr>
<tr>
<td>38°</td>
<td>37.95</td>
<td>1514</td>
</tr>
<tr>
<td>39°</td>
<td>38.97</td>
<td>1514</td>
</tr>
<tr>
<td>40°</td>
<td>39.98</td>
<td>1514</td>
</tr>
<tr>
<td>41°</td>
<td>41.00</td>
<td>1514</td>
</tr>
<tr>
<td>42°</td>
<td>41.96</td>
<td>1514</td>
</tr>
<tr>
<td>43°</td>
<td>42.97</td>
<td>1514</td>
</tr>
<tr>
<td>44°</td>
<td>43.98</td>
<td>1514</td>
</tr>
<tr>
<td>45°</td>
<td>45.00</td>
<td>1514</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Angle</th>
<th>a-value</th>
<th>b-value</th>
</tr>
</thead>
<tbody>
<tr>
<td>31°</td>
<td>30.37</td>
<td>1417</td>
</tr>
<tr>
<td>32°</td>
<td>32.01</td>
<td>1514</td>
</tr>
<tr>
<td>33°</td>
<td>33.05</td>
<td>1514</td>
</tr>
<tr>
<td>34°</td>
<td>34.02</td>
<td>1514</td>
</tr>
<tr>
<td>35°</td>
<td>34.99</td>
<td>1514</td>
</tr>
<tr>
<td>36°</td>
<td>35.96</td>
<td>1514</td>
</tr>
<tr>
<td>37°</td>
<td>36.93</td>
<td>1514</td>
</tr>
<tr>
<td>38°</td>
<td>37.95</td>
<td>1514</td>
</tr>
<tr>
<td>39°</td>
<td>38.97</td>
<td>1514</td>
</tr>
<tr>
<td>40°</td>
<td>39.98</td>
<td>1514</td>
</tr>
<tr>
<td>41°</td>
<td>41.00</td>
<td>1514</td>
</tr>
<tr>
<td>42°</td>
<td>41.96</td>
<td>1514</td>
</tr>
<tr>
<td>43°</td>
<td>42.97</td>
<td>1514</td>
</tr>
<tr>
<td>44°</td>
<td>43.98</td>
<td>1514</td>
</tr>
<tr>
<td>45°</td>
<td>45.00</td>
<td>1514</td>
</tr>
</tbody>
</table>
The statement of angles is always programmed from the quadrant start.

Thus, the a,b values may have X,Y and Z characteristics.

**Exercise:**

Put in the a,b values of quadrants IV and I.

### Radius 10 mm

<table>
<thead>
<tr>
<th></th>
<th>IV</th>
<th>I</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td></td>
<td></td>
</tr>
<tr>
<td>b</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Radius 27 mm

<table>
<thead>
<tr>
<th></th>
<th>IV</th>
<th>I</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td></td>
<td></td>
</tr>
<tr>
<td>b</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Exercise:

Put in the coordinates for $P_O$, $P_A$, $P_Z$ and $P_E$.

Radius 10 mm

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_O$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_A$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_Z$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_E$</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Radius 38 mm

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_O$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_A$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_Z$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_E$</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Programming of Arcs ≠ 90° in absolute Mode

For a better understanding some details on the F1-CNC computer:
In the memory (RAM) the 90° arcs (Quadrants) are stored -
with the block:
N.../G02/X=1500/Y=1000/Z....

The computer knows
- sense of rotation (G02)
- position and size of the 90° arc
  (statement of coordinates of end point PE of 90° arc).

The starting coordinate PO of the 90° arc is known to the computer from the previous block.

In the computer, this quadrant is divided into 90 steps of 1° each.

Manufacture of the 90° arc

The computer instruction is:
Traverse all 90 steps of the programmed quadrant.
Programming of Arcs from 0° to $\alpha \neq 90°$

We instruct the computer to edit only a part of the 90 steps.

This is done with the M99 information

$J=0$ to $K=30$

Flow in the computer

N99/G01/X=0/Y=500/Z......
N100/G02/X=1500/Y=1000/Z......
N101/M99/J=0/ K=30

1. The computer checks whether starting and end coordinates of the 90° arc are correct.

   It compares the coordinates of blocks N99 and N100.

2. The computer asks whether there is a M99 instruction in the following block.

<table>
<thead>
<tr>
<th>No</th>
<th>Yes</th>
</tr>
</thead>
<tbody>
<tr>
<td>All 90 steps are edited</td>
<td>- It calculates (&quot;theoretically&quot;) all steps up to J.</td>
</tr>
<tr>
<td></td>
<td>- It edits traverse instructions from J to K</td>
</tr>
<tr>
<td></td>
<td>- It calculates from K to 90° without editing instructions</td>
</tr>
</tbody>
</table>
Programming $\alpha \neq 0^\circ$ to $\alpha = 90^\circ$ in absolute Mode

1. Programming to point PA
   
   \[ N100/G01/X616/Y468/Z \ldots \]

2. Arc = 28° to 67°

2.1. Description of the 90° arc:
   
   \[ N101/G02/X1616/Y1468/Z \ldots \]

   The absolute coordinates of the quadrant end point PE are described starting from point PA.

   By computation this is the end point of the quarter arc.

   \[
   XE = XA + \frac{a}{R} \\
   YE = YA + \frac{a}{R} \\
   ZE = ZA
   \]

2.2. N102/M99/J28/K67

Flow of data in the computer - Manufacture

1. The computer checks whether coordinates of starting point PA and quadrant end point PE are correct (absolute).


   a) Computer proceeds up to J28 (= 28°) - without traverse instruction.

   b) It gives traverse instructions from J28 to K67 (28°-67°). The impulses from J28 to K67 are worked through. The indicated quadrant is manufactured - from starting point PA to target point PZ.
A Method of programming Arcs \( \alpha \neq 90^\circ \) (absolute)

With partial arcs \( \alpha \neq 90^\circ \) it is often necessary to calculate starting and target point of the previous and the following blocks: thus it is useful to establish a chart.

**Specification:**

PA - Starting point of partial arc of circle

PZ - Target point of partial arc of circle

PE - End point of quadrant ("theoretical" target point)

PO - Starting point of quarter arc.

**Examples:**
PA: PA is the target point of the block before the circle programming.

XA
YA
ZA

PE: "Theoretical" end point of the quarter arc
XE = XA + R
YE = YA + R
ZE = ZA (interpolation in the plane)

PZ: Programmed target point
XZ = XA + Δ X
YZ = YA + Δ Y
ZZ = ZA (interpolation in the plane)

Coordinates path of the partial radius
Δ X = XPZ - XPA
Δ Y = YPZ - YPA
Δ Z = 0 (interpolation in the plane)

PO: Theoretical starting point of the quarter arc
XO = XA - a
YO = YA - b
ZO = ZA
Exercise:

Put in X,Y-values, Z-value = 0

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>PA</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PE</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PZ</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PO</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Program the path

W−PA−Pz−P1
Exercise:

Put in X,Y-values, Z-value = 0

Program path
W - PA - Pz - P1

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>PA</td>
<td>X</td>
<td>Y</td>
</tr>
<tr>
<td>PE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Po</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>


<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L)</th>
<th>remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Exercise:**

Slot 3 mm deep

Programming: in absolute mode

Zero point of workpiece as in drawing.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J) (D)</th>
<th>Y (K) (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
<th>remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
G04 - Dwell

If you manufacture a borehole and withdraw the drill after you have reached the desired depth, then the chip will be torn off. The base of the borehole has steps.

With boreholes of tapered shape this often does not matter. With shouldered boreholes, however, it can be disturbing.

The same applies for milling cutters of larger diameter or for fly wheel cutter if you move away suddenly.

You have an unwanted shoulder in the workpiece.

In such cases a dwell should be programmed.

Programming

Format G04
N3/G04

The tool remains 0,5 seconds in the programmed position of the previous block.
G21 – Empty Line

You may program as many empty lines as you wish in a program. The empty lines are jumped over in the program sequence. In the place of empty lines you can program at later stage other G- or auxiliary functions.

Format G21
N3/G21
Subroutines
G25/M17

The subroutines are "managed" by the main program.

In the main program the movements are programmed up to the starting point for the subroutines.

MAIN PROGRAM

At the end of a subroutine the instruction is given to carry on with the main program.
Subroutines

It happens quite often that various milling operations of same shape are manufactured at one and the same workpiece.

Example

- 4 geometrically identical pockets.
- For the manufacture of each pocket the milling cutter has to be moved to working position.
- The programming and manufacturing process is the same for each individual pocket. You program in one program pocket milling for 4 times.

These identical operations may be programmed just once and then "stored". If they are needed they are called up

To our example

1. The tool is moved to the first milling start point.
2. The subroutine is called up. The first pocket is being milled.
3. The tool is then moved to the second milling start point.
4. Subroutine is called up.
5. The tool is then moved to the third milling start point.
6. Subroutine is called up.
   etc.
Principle: Call-up of Subroutine and Sequence on F1-CNC

MAIN PROGRAM

UP 1

UP 2
Subroutine-Programming
G25 Jump to Subroutine
M17 Jump back to Main Program

1. Programming up to the first start of the subroutine (assume NO5).

2. Call up subroutine G25 in block NO6:
   NO6/G25/L100
   - With G25 the subroutine is called up.
   - Under the F-address we describe the block number with which the subroutine begins.
     In this case the subroutine begins with block no. N100 (the block no. is selected by the programmer).

3. The subroutine:
   N100/
   N101 ...
   N102 ...
   N103 ...
   N104 ...
   N105/G01
   In the subroutine the operation to be repeated is described (block N100 to block N105)

4. Jump back instruction M17:
   At the end of a subroutine you have the jump back instruction M17. The program jumps to the following block with which the subroutine was called up.
Example

- Programming main program: absolute
- Programming subroutine: incremental
- Zero point of workpiece as in drawing
- Reference point set-off as in drawing
- Diameter of milling cutter 8 mm

Continue the program. Start point shall be end point of program.

In block NO5 the workpiece zero-point is programmed again.
More Subroutines

You can write as many subroutines in a program as you like.

Example

The slots 1 + 2 are subroutine no. 1. The slots 3 + 4 are subroutine no. 2. The program shows an incremental main program.
Example
You can use block N105 to N106 for the manufacture of slot 1 and 2.

An example:
- Slot 1 and slot 2 are identical and contained in cross slot 3 and 4.
- You write a subroutine for slot 1 and 4.
  N100/G91
  N101/G01 to
  N108
  N109/M17
You can use block N105 to 108 for the manufacture of slot 1 and 2.

It is possible to call up parts of a subroutine.
In this example:
Block N105 to N109 / M17
Part of a subroutine program

The scheme shows an incremental main program. In an absolute main program you have to determine the workpiece zero-point with G92.

NCC
:
:

NO5 Milling cutter is positioned for subroutine

NO6 / G25 / L100
NO7 / G00
NO8 / G25 / L100
NO9 / G00
N10 / G25 / L105
N11 / G00
N12 / G25 / L105

Slot 1

N100
N101
N102
N103
N104
N105
N106
N107
N108
N109 / M17

Slot 2

Slot 3

Slot 4

Jumps back
Example G25/M17

Program this example:

Width of slot 6 mm
Depth of slot 5 mm
Zero point of workpiece as in drawing

Decide yourself between absolute or incremental value programming.

Start point as in drawing.

<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>(M)</th>
<th>X:</th>
<th>(D)</th>
<th>Y:</th>
<th>(S)</th>
<th>Z</th>
<th>F:</th>
<th>(L)</th>
<th>(T)</th>
<th>(H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Example:
You have to mill a rectangular slot. Since the slot is deep you need a few runs; these are identical in the XY-plane.

Example:
- Mill cutter is already cutting at block no. N005.
- N006 is jump to subroutine.
- The subroutine consists of block N101 to N105.
- N105 is jump back to main program.
- N007 is infeed in main program.
- N008 is jump to subroutine.
  etc.
Exercise

Program the workpiece. The depth of cut shall be reached in 3 runs.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L/T/H)</th>
<th>remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Exercise

- Make a sketch indicating the start point.
- Determine the zero point (W)
- Main program: absolute
- Circular slot in 2 runs

Depth of slot 10 mm
The Nesting of Subroutines

Call-up – Sequence
G27 – Jump Instruction
Format N3/G27/L3

With this instruction you can jump forward or backward within the program.
- Under the L(F) address the block is programmed until to the one where the program shall be skipped.

Example
Block 17
Instruction to jump to block 110

Block 120
Instruction to jump back to N18

Application
- The surface of the workpiece shall be worked or not.
- You describe a finishing program (N4 to N12).
- In the block proceeding the finishing operation you program G21.
- In blocks N4 to N12 the finishing cut is carried out.

Jump instruction
If the surface should remain unfinished:
Delete N3/G21
Program N3/G27/L13
The blocks N4 to N12 are skipped.
The Cutter Radius Compensation –
Parallel to Axis

G40 – Cancel the compensation
G45 – Add cutter radius
G46 – Deduct cutter radius
G47 – Add cutter radius twice
G48 – Deduct cutter radius twice

G45/G46/G47/G48 are self-maintaining functions. They are revoked by G40 or M30 (program end). G45 can be overwritten by G46/G47/G48 and vice-versa.

Before programming G45/G46/G47/G48 you have to describe the tool data under M06.

In examples up to now we have always been programming the center line path of the cutter. With the lengths to be worked the cutter radii had to be added or deducted. This calculation work can be taken over by the computer, if appropriate informations are given.
G45 – Adding Milling Cutter Radius

Programming incremental

The milling cutter shall touch the inside of the contour.

Conventional programming:
N.../G00/X=l+r/ ....
The radius has to be added to the length l.

Programming with G45 (Adding Cutter Radius)

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (L)/(T)/(H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

1. The computer has to know the cutter radius so that it can calculate the correct movement (l + r).

In one of the previous blocks the tool data have to be described, otherwise alarm sign A18.

2. Call up G45: 
Add cutter radius once.

... M06 D500 S2000 0 T01

3. Program movement.
Measure L (30)

The computer picks up the tool data from the M06 instruction which was programmed last.

... M06 D500 S2000 0 T01

G45

... 00 3000 0 0

Cancel the cutter radius compensation:
N.../G40
**G46 – Deducting the Cutter Radius**

Mode of programming: incremental

Cutter shall touch outer contour.
Cutter dia. 10 mm

**Programming:**

```
N100/M06/D500/S2000/Y=0/T(F)1
N101/G46
N102/G01/X=L/Y=0/Z=0/F...
```

The cutter moves by the distance \( l - D \).

Approaching an Edge – Not parallel to Axis

Programming: incremental

Cutter dia. 16 mm
Reference dimension \( H_z = 0 \)

```
N01/M06/D800/S1700/Y=0/T(F)1
N02/G46
N03/G01/X4000/Y2000/Z=0/F...
N04/M30
```

Approaching an Edge – Not parallel to Axis

Programming: absolute

- Cutter dia. 16 mm
- Zero-point as in drawing

```
N00/G92/X-4000/Y-3500/Z1000/
N01/M06/D800/S2000/Z=0/T01
N02/G46
N03/G00/X=0/Y=0/Z1000
N04/M30
```
Exercises G45/G46

- Program the distance/traverse $P_1 - P_2$ in absolute and incremental mode.

- Radius $D$: 12 mm

- Zero-point from point $P_1$. 


<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (L)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (L)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (L)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
G47 – Add Cutter Radius Twice

- Outside contour shall be milled
- Mode of programming: incremental
- Cutter radius 6 mm
- Starting point as in drawing

**Programming:**

```
N000/M06/D600/S2000/Z=0/T(F)1
N1/G46
N2/G01/X2000/Y1500/Z=0/F...
N3/G47
N4/G01/X4000/Y=0/Z=0/F...
N5/G01/X=0/Y3000/Z=0/F...
N6/G01/X -4000/Y=0/Z=0/F...
N7/G01/X=0/Y -3000/Z=0/F...
N8/G46
N9/G00/X -2000/Y -1500/Z=0/
N10/M30
```

Block N4 to N7

Cutter radius is added twice.

Block N02, N9

Cutter radius is deducted once.

**Cutter path plotted**
Programming exercise:

Cutter radius 5 mm

Incremental programming
Starting from point P₁.

Absolute programming
Determining the zero-point starting from point P₁.
G48 – Deduct Cutter Radius Twice

Example: Milling an inside contour

- Milling cutter radius 6 mm
- Mode of programming: incremental

Program:

Block N3: move in
Block N5 - N8: inside contour
Block N9: move out of inside contour
Block N11: withdrawal to starting position

Cutter path plotted in one plane
Exercise:
Cutter radius 5 mm

Incremental programming
Starting from point P₁

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L/T/H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Absolute programming
Determining the zero-point from point P₁.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L/T/H)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Example: Combined Inside-/Outside Contour

Mode of programming: incremental
Milling cutter radius 5 mm

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J) (D)</th>
<th>Y (K) (S)</th>
<th>Z</th>
<th>F</th>
<th>remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>M06</td>
<td>0 500</td>
<td>5 2000</td>
<td>0</td>
<td>T01</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>46</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>00</td>
<td>3.500</td>
<td>1 000</td>
<td>0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>47</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>01</td>
<td>0</td>
<td>2 000</td>
<td>0</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>40</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>01</td>
<td>2.500</td>
<td>0</td>
<td>0</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>48</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>01</td>
<td>0</td>
<td>2 000</td>
<td>0</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>40</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>01</td>
<td>-2.500</td>
<td>0</td>
<td>0</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>47</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>01</td>
<td>0</td>
<td>2 000</td>
<td>0</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>01</td>
<td>5 000</td>
<td>0</td>
<td>0</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>01</td>
<td>0</td>
<td>-6 000</td>
<td>0</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>01</td>
<td>-5 000</td>
<td>0</td>
<td>0</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>46</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>00</td>
<td>-3.500</td>
<td>-1 000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>M30</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Exercise:
- Program the example in absolute mode.
- Zero point as in drawing.
- Cutter radius 5 mm
G64 – Switching Feed Motors Currentless

The previously programmed G- and M-codes remain stored.

Switching currentless with program stored

G64 is a pure switching function. It is not stored.

1. Press key [→] until G-lamp flashes.
2. When a number appears on the VDU, press key [DEL].
3. Key in [6 4].
4. Press key [1 INP], the feed motors are now currentless.
G72 – Pocket Milling Cycle

Pockets are a quite common shape when milling. The programming work of many single blocks can be put together to a cycle. The computer offers a fixed sequence = cycle.

Programming G72

1. G72
2. X-value, inside dimension of the pocket in X-direction.
4. Z-value = depth of pocket
5. F-value

Format G72

\[ \text{N3/G72} / \pm 5/Y \pm 4/Z \pm 5/F3 \]

With this block the machine cannot mill a pocket yet.
- It does not know the radius of the cutter and thus cannot calculate the movements.
- Therefore, the tool has to be described in one of the previous blocks (M06).

The computer uses these data (cutter radius) to calculate the effective movements which were programmed last.

If no M06 was programmed before, alarm sign 15 will appear.
Pocket Milling Sequence

The milling cutter has to be positioned before the pocket milling can start.

1. The cutter moves into the pocket by the $z$-value, if a $z$-movement is programmed.

2. Milling out (reaming) a pocket:
   - The first movement is in $x$-direction.
   - The signs determine the sequence of the traverse.

Overlap:

The overlap is $1/10$ of the cutter radius (with 5 mm radius approx. 0.5 mm).

The computer takes the information about the radius from the MO6 block which was programmed last.
3. Finishing ram:
The sides are being finished. Traverse 10/11/13. Finishing measure approx. 1/10 of the cutter radius.

4. Cutter moves out of pocket (Z-direction) into starting position.
The pocket milling cycle is complete.

Pockets can be programmed in absolute or incremental mode.

Incremental programming:
X, Y, Z values are given from the starting position.

Technological tip
When moving in a milling cutter the feed should be approx. halve of the normal cutting feed.
Therefore it is advisable to program this first movement in an extra block.
Summary G72 (M06)

With pocket in XY-plane

Data for calculation of cutter path

\[ \text{N.../G72/X} \quad /Y \quad /Z \quad /F \]

X-value  Y-value  Z-value

- M06
- D(X) = Cutter radius
- S(Y) = Speed
- Z = HZ-value
- T(F) = Tool number

X = Inside measurement of pocket
Y = Inside measurement of pocket
Z = Infeed depth
F = Feed

The computer will calculate all reference points automatically.

Example:

- Cutter diameter 10 mm
- The pocket is programmed incrementally
- Start position for cycle as in drawing.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L) (T) (M)</th>
</tr>
</thead>
<tbody>
<tr>
<td>05</td>
<td>00</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>06</td>
<td>M06</td>
<td>500</td>
<td>2000</td>
<td>0</td>
<td>01</td>
</tr>
<tr>
<td>07</td>
<td>72</td>
<td>4000</td>
<td>3000</td>
<td>-500</td>
<td></td>
</tr>
</tbody>
</table>

N5 = Move to start position
N6 = Tool data
N7 = Pocket milling cycle
Example:
- Cutter diameter 8 mm
- Programming mode: incremental

Example:
- Programming mode: absolute
- Determine the zero point of the workpiece
- Mill the pockets in two runs with two subroutines, if you know G27 already.
Boring

With G00 and G01 you can execute boring operations:

1. You program with G01 (feed) at desired depth of bore

2. With rapid traverse you move to the starting point of the boring operation.

The procedure is always the same one:
- Boring with feed (G01) to length L
- Withdrawal by length L with G00.

Therefore these two movements are put together in one G-function (cycle).

G81 – Boring Cycle

Programming:
N.../G81/Z.../F...

Under the Z-address you program the depth of bore.
F-address: feed in mm/min
The withdrawal is done automatically with G00.

Format G81
N3/G81/Z ± 5/F3

Application:
Through holes with a not too large depth of bore.
G82 – Boring Cycle with Dwell

If the depth of bore is reached, the withdrawal with G81 starts immediately (rapid traverse). The bore chip is torn off. - The surface at the base of the hole is not clean.

Therefore the drill bit remains in the programmed position Z.

Sequence

1. First movement: with feed
2. If depth of bore is reached, the drill bit turns without feed 0.5 seconds.
3. Withdrawal in rapid traverse.

Programming:
N.../G82/Z±....../F...

Format G82
N3/G82/Z ± 5/F3

Application:
Blind holes of medium depth.
G83 – Withdrawal Cycle

- It happens quite often with deep bores that the chips are not flowing out properly.

- Therefore you have to withdraw the drill bit in order to take away the chips.

You can program the operation with G01/G00/G01/G00 etc. or with various G81 or G82 cycles.

The drawing illustrates the principle, that a few cycles are again put together to a new cycle.
Programming G83:

N.../G83/Z±....../F...

The final depth of bore and the feed are to be programmed.

Procedure:

1. Bore at 6 mm depth with feed
2. Withdrawal with rapid traverse (6 mm)
3. With rapid traverse 5.5 mm and 6 mm feed
4. Go to starting point with rapid traverse
5. With rapid traverse 11 mm, with feed 6 mm etc. until you reach the programmed Z-value.

Format G83
N3/G83/Z±5/F3

Application: Deeper bores
Example:

Pay attention to the technological data.
Use drilling emulsion to protect the drill bit.
Bores larger than 10 mm dia. need to be rough-drilled.

Use G81, G82, G83.
G85 – Reaming Cycle

In order to achieve bores with a high surface quality, reaming of bores is necessary.

Using standard twist drill you may reach quality 11 to 12. For higher quality standards the bores have to be reamed. By reaming you reach quality 6.

G85 is a combination of two G01 commands.

Programming:
- Block number
- G85
- Z-value
- Feed F

Format G85
N3/G85/Z±5/F3

Note:
The depth of the bores to be reamed is indicated in the technical drawing. The bore-length 25 has a tolerance measurement.
G 89 - Reaming Cycle with Dwell

The sequence is the same as with G85. The reamer bit remains 0.5 seconds in the dead position if the programmed depth is reached.

Sequence

Infeed with feed at length Z

0.5 seconds dwell

Withdrawal with feed at length Z

Format G89

N3/G89/Z ± 5/F3
Chapter 6

Tools, tool lengths compensation, radius compensation of milling cutter

Programming of tools 6.1
Tool lengths compensation (principle) 6.3
Working with various tools 6.5
1. Determining the tool sequence 6.7
2. Determination of tool data
   2.1. Diameter, technological data 6.7
   2.2. Detecting the tool length differences 6.9
3. Calculation of tool lengths 6.13
4. Tool lengths compensation in the program sequence 6.15
5. Tool lengths corrections 6.17—6.21
Other cases for programming M06 6.23
Connection: Zero-point offset G92
Tool lengths compensation M06 6.25
Milling of chamfers 6.27—6.33
Depth of bore with spiral drill 6.35
Tool data sheets
Tool sheets
The Programming of the Tools

Tool magazines of industrial NC-machines are equipped with up to 50 or more tools.

The sequence is programmed. Technological data and dimensions have to be programmed for each individual tool bit.

Tools are programmed using the T-address. T stands for tool.
Tool Lengths Compensation

The computer is given information on the target position or desired position.

Imagine the coordinate system transferred into the reference plane of the tool.

The target position is described starting from the actual position.

T01
[M06/D.../S.../Hz = O/T01]

T02
[M06/D.../S.../Hz = +.../T02]

T03
[M06/D.../S.../Hz = -.../T03]
Working with various Tools

Determining the tool sequence
Detecting the tool data
Compensation of tool lengths

For the manufacture of a workpiece you often need different tools: drills, various milling cutters etc.

The programmer needs to know various data such as
- kinds of tools
- application of different tools,
- position of tools to each other

1. The milling cutters are of different diameters. These are known to you.

2. The tools are of different lengths. These are not known to you. You have to measure the lengths and take them into consideration when programming. Otherwise you move the cutter in the air without chip removal or you run it into a workpiece (crash).
Procedure

1. Determining the tool sequence

2. Determination of tool data
   2.1. Diameter, technological data

<table>
<thead>
<tr>
<th>T1</th>
<th>T2</th>
<th>T3</th>
<th>T4</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Tool T1]</td>
<td>![Tool T2]</td>
<td>![Tool T3]</td>
<td>![Tool T4]</td>
</tr>
</tbody>
</table>

Entering the data

1. Stick the tools into the corresponding column.

2. Enter the technological data:
   - \( d \) = Cutter diameter
   - \( D \) = Cutter radius
   - \( F \) = Speed of feed
   - \( t \) = Maximum depth of cut
   - \( S \) = Speed

These data will make the programming easier.
2.2. Detecting the Tool Length Differences (H₂)

The differences in tool lengths have to be measured. The measurements can be taken using an external presetting device. In many cases the measuring system within the CNC-machine is taken use of.

You can scratch with all tools a reference surface or measure the data using a dial gauge.

The difference is called H₂.

Procedure

Mount T₁ (reference tool) and scratch reference surface, set dial gauge respectively.

Detection of data by scratching

Scratching only when cutter is turning

Detection of data with dial gauge.

Set dial gauge when machine is at stand-still.

Press key DEL, the Z-value display is set to 0.
Mount T2

Scratch surface

Touch dial gauge with cutter until it shows 0.

Read value from display.

Enter value into tool data sheet. In this way you determine all tool lengths.

Pay attention to the signs!
3. Calculation of Tool Lengths  
(Tool lengths compensation)

Since these data are known you could take the various lengths into consideration. This would, however, be quite confusing calculation work and will often lead to mistakes.

Calculation of tool length M06  
(Tool lengths compensation)  
(Programming)

The data are entered into the programming sheet.

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (T)</th>
</tr>
</thead>
<tbody>
<tr>
<td>M06</td>
<td>2000</td>
<td>1100</td>
<td>0</td>
<td>4</td>
<td>1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (T)</th>
</tr>
</thead>
<tbody>
<tr>
<td>M06</td>
<td>500</td>
<td>2000</td>
<td>650</td>
<td>2</td>
<td></td>
</tr>
</tbody>
</table>

T = tool number
D = milling cutter radius
S = spindle speed (only for your information)
$H_Z =$ difference in tool length

Note:

If you write a number 1, 2, 3, 4 under the $F(T)$ address when programming M06, this automatically means program hold.
If there is a 0 under the $F(T)$ address, there will be no hold.
Tool Lengths Compensation in the Program Sequence

The first tool (T01) has a \( H_z \) value = 0.

\[
N.../MO6/D2000/S1300/Z(H_z) = 0/T01
\]

\[
\downarrow\quad \text{Manufacture}
\]

\[
\downarrow\quad \text{Tool change T02}
\]

\[
N.../MO6/D500/S2000/Z(H_z) = 800/T02
\]

\[
\downarrow\quad \text{Start}
\]

First the tool T02 moves from the actual position to the target position.

Then the manufacture itself starts.
You have finished the manufacture of a workpiece and find out that the Z-measurement is not correct.

- The program is correct
- The starting position of the cutter is correct.

What is the reason?

The target value information ($H_z$ value) was not correct (wrong, inaccurate measurements, cutter not resharpened).

**TARGET INFORMATION $H_z$ wrong**

$MO6/D.../S.../Z+ 12.43/T02$

The target information $H_z$ has to be corrected.

$H_{zk} = \text{Corrected target information}$

$H_{zk} = H_z + (\Delta Z \text{ correction value})$

$MO6/D.../S.../Z+ 1100/T02$
Example of a Correction of the Hz-value

You may

1. Measure tool once again
2. Detect the correction value by measuring the workpiece.

The Hz information has to be corrected by the \( \Delta Z \) value.

- Imagine the coordinate system transferred to the \( Z \)-actual position of the workpiece.
- Add the correction value \( \Delta Z \) to the target information \( Hz \) of the tool bit.

Pay attention: \( \Delta Z \) may have \(^\pm\) sign.

\[
Hz_k = Hz + (\pm \Delta Z)
\]
\[
= 15.4 + (-1.35)
\]
\[
= 14.05
\]

The value \( Hz_k = 14.05 \) is corrected in the programming sheet, tool data sheet and in the memory.
Example

Programmed Hz-value (actual information): 
- 6,25 mm

Workpiece measurements: Actual and target, compare drawing.

Correct the Hz-value

$$Hz_{k} = Hz + (\pm \Delta Z)$$

Pay attention to the sign of $$\Delta Z$$.

$$Hz_{k} = \ldots \ldots$$

Example

Hz of TO1 = 0
Hz of TO2 = -4,32

Workpiece:

Actual value TO1 = 10,5 mm
Actual value TO2 = 5,2 mm

Target value TO1 = 10 mm
Target value TO2 = 6 mm

Correct the Hz-values of TO1 and TO2.
Other Cases for Programming M06

If a G45, G46, G47, G48 or a G72 command (cutter radius compensation) is programmed, in one of the previous blocks a M06 has to be put in, otherwise the alarm sign will appear.

A16: Cutter radius information missing

The computer needs the cutter radius information D in order to calculate the compensated paths (G45,G46,G47,G48).

The same applies with the pocket milling cycle G72.

Alarm A16

Cutter radius information missing.
Connection:
G92 Zero-point offset
M06 Tool lengths compensation

The Hz-information is an incremental target information within an independent coordinate system.

The origin of the coordinate system is determined with G92.
Milling of Chamfers

Chamfers are usually milled at an angle of 45°.

The size of the chamfer is determined by the programmed path and/or by the cutting contour.

1. Chamfer size determined by different cutter paths (different distances between cutter axis and workpiece edge)

2. Chamfer size determined by different infeed and Z-direction. The cutter path remains unchanged.
Programming a Chamfer
with Cutter Path unchanged

The contour is milled with a cutter of 10 mm dia.
To avoid the necessity to program a new cutter path for chamferring, the angle cutter shall be programmed in Z-direction such that a chamfer 1x1 mm is reached.

Cutter path - end mill
Cutter path - angle cutter

How deep has the Angle Cutter to be fed in?

The radius of the angle cutter which mills the inside contour of the chamfer:

\[
\text{Radius end mill} + \text{Width of chamfer}
\]

With a mill path using a 5 mm shank, dia. 6 mm, the radius of the angle cutter produces the chamfer 1x45°.

\[
R = \frac{5}{\tan 45°} + \frac{1}{\tan 45°}
\]
Angle cutter, dia. 16 x 4 mm

With a 45° angle cutter, the cutting radius changes by one mm if the cutter is fed in by 1 mm.

Example

Radius of mill path 5 mm

1. Cutter at height 0
Distance to workpiece = 1 mm

2. Cutter fed in by 1 mm
Radius 5 mm touches edge.

3. Cutter fed in by 2 mm
Chamfer 1x45° is produced.
Measure of total depth:
Measure until radius mill path (1 mm) +
Width of chamfer (1 mm)

= 2 mm
Example

Unchanged mill path
- Radius end mill: 5,63 mm
- Chamfer 0,67 x 0,67 mm

With an infeed of 1,63 mm the angle cutter touches the contour.

- Infeed 1 mm R5
- Infeed 1,63 mm R5,63

Radius 6,3 mm produces the chamfer contour.

\[
5,63 \text{ mm radius cutter path} + 0,67 \text{ mm width of chamfer} = 6,30 \text{ mm}
\]

Cutter infeed

- 1,63 mm (radius touches contour)
- 0,67 mm (width of chamfer)
- 2,30 mm total infeed
The Depth of Bore with Spiral Drill

Blind holes are dimensioned down to the flat ground of the bore.

If you want to calculate the tool length you either scratch the surface with the point of the drill bit or you take measurement of the length of the tool.

In order to program the indicated depth of bore you have to add the length of the tool point.

\[ \tan 30^\circ = \frac{H}{\frac{d}{2}} \]

\[ H = \tan 30^\circ \times \frac{d}{2} \]

<table>
<thead>
<tr>
<th>Drill dia. in mm</th>
<th>H (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.57</td>
</tr>
<tr>
<td>4</td>
<td>1.15</td>
</tr>
<tr>
<td>6</td>
<td>1.73</td>
</tr>
<tr>
<td>8</td>
<td>2.30</td>
</tr>
<tr>
<td>10</td>
<td>2.89</td>
</tr>
<tr>
<td>12</td>
<td>3.46</td>
</tr>
<tr>
<td>14</td>
<td>4.04</td>
</tr>
<tr>
<td>16</td>
<td>4.61</td>
</tr>
</tbody>
</table>

Drill Data for the Tool Sheet

Always deduct value \( H \) from the measured data when you enter it. You need not to calculate anymore and can program the dimensions of the drawing directly.
# Tool Data Sheet

<table>
<thead>
<tr>
<th></th>
<th>T1</th>
<th>T2</th>
<th>T3</th>
<th>T4</th>
<th>T5</th>
<th>T6</th>
<th>T7</th>
<th>T8</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th></th>
<th>d</th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>$\frac{d}{2}$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>F</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>t</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>S</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HZ</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HZK</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>(mm)</th>
<th>Cutter dia.</th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>(mm)</td>
<td>Cutter radius</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>F</td>
<td>(mm/min)</td>
<td>Feed speed</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>t</td>
<td>(mm)</td>
<td>Max. milling depth</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>S</td>
<td>(U/min)</td>
<td>Spindle speed</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HZ</td>
<td>(mm)</td>
<td>Difference measure</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HZK</td>
<td>(mm)</td>
<td>Corrected difference measure</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **Vertical axis system**
- **Horizontal axis system**

<table>
<thead>
<tr>
<th></th>
<th>+Z</th>
<th>+Y</th>
<th>+X</th>
<th>+Z</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Zero-point of workpiece</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start position</td>
</tr>
<tr>
<td>Tool change position</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Zero-point offset (G92)</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
</tr>
<tr>
<td>Y</td>
</tr>
<tr>
<td>Z</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Drawing no.:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Denomination:</td>
</tr>
<tr>
<td>Workpiece material:</td>
</tr>
<tr>
<td>Program no.:</td>
</tr>
<tr>
<td>Name:</td>
</tr>
<tr>
<td>Date:</td>
</tr>
</tbody>
</table>
Chapter 7
The M-Functions
The M-Functions
Miscellaneous or switching functions.

M00 – Program Hold

If you program M00 in a block, then
the program will be interrupted.

Continuation of the program: press
START key.

When Do We Program M00?
- Tool change
- Take measurements
- Switch to hand operation
- Carry out corrections
etc.

M30 – Program End

In the last block of a program you
have to program M30. Otherwise the
alarm sign A05 will appear.

After M30 the program jumps automati-
cally to NOO. You can start anew.

If the DNC interface is mounted, M30
switches off the main spindle (M03
is cancelled).
**M03 — Milling Spindle on**

*only with accessory DNC-Interface*

The M03 instruction switches on the milling spindle. Switch the milling spindle on such that the motor has enough time to run up and that you are in position to set the right rpm.

**Important note M03**
Before pushing the start key the main spindle switch has to be set to CNC-position.

**Format M03**

```
N3/M03
```

---

**M05 — Milling Spindle Off**

*only with accessory DNC-Interface*

**Format M05**

```
N3/M05
```

When do we program M05?
- Before a tool change
- Before taking measurements

**Note:**

M30 switches off the milling spindle too.
M06 switches off the milling spindle if T(F) ≠ 0.
M06 – Tool Lengths Compensation

Format M06

N3/M06/D5/S4/Z(Hz) ± 5/T(F)3

Compare chapter "Tool Lengths Compensation"

M17 – Jump Back Into Main Program

Format M17

N3/M17

Compare "Subroutines"

M99 – Circle Parameter

Format M99

N3/M99/J2/K2

Compare "Circle Programming"

M08, M09, M20, M21, M23, M26 are as switching functions not yet defined.

With them you could activate peripheral devices (under preparation!)
Chapter 8
Input of Program, Corrections, Operation

Survey \[8.1\]
What happens when data is put in? \[8.2-8.3\]
Input format \[8.4\]
Indication on the screen \[8.5\]
Input of program \[8.6-8.7\]
Operating elements CNC; Program input \[8.9\]
Option key hand operation — CNC operation \[8.9\]
The word indication \[8.10\]
The figure keys, the minus key \[8.12\]
The memory key[LINP] \[8.13\]
The \(\rightarrow\) key \[8.14\]
The \(\text{FWD}\) key \[8.15\]
The \(\text{REV}\) key \[8.16\]
The \(\text{DEL}\) key \[8.17\]
Input of M-values \[8.18\]
Take-over of registered values \[8.19\]
Inserting and deleting of blocks \[8.20\]
Deleting of a registered program \[8.21\]

Program Sequence \[8.23\]

Testrun \[8.25\]
Single block operation \[8.26-8.27\]
Automatic operation \[8.29\]
Interventions during program flow \[8.31-8.33\]
— Program stop
— Program hold
The knobs, displays, symbols, etc. will confuse you in the beginning. So first put in the very simple programs and check the various function keys. In half an hour you will be accustomed to them.
Survey

Data Input, Correction, Delete

Storing a word

\[ \text{INP} \]
Take over of values

Correcting a word

\[ \text{DEL} \rightarrow \boxed{\text{Put in value}} \rightarrow \text{INP} \]

M-programming

Press M

Searching a word

\[ \rightarrow \]

Searching a block

\[ \text{FWD} \rightarrow \text{REV} \]

Inserting a block

\[ \rightarrow + \text{INP} \]

Deleting a block

\[ \rightarrow + \text{DEL} \]

Deleting a program

\[ \text{DEL} + \text{INP} \]
(first DEL)

Set program to NOO

\[ \text{INP} + \text{REV} \]

Sequence of Program

Testrun:

Inching through the program with M

Single block operation

\[ 1 + \text{START} \]
\[ 2 + \text{_ _ _ _ _ _} \]
\[ 3 + \text{_ _ _ _ _ _} \]
(first number key)

Automatic operation

\[ \text{START} \]

Influencing the Program

Termination

\[ \text{INP} + \text{REV} \]

Interruption

\[ \text{INP} + \text{FWD} \]

Storing of Program

Compare tape operation

\[ \text{RS-232 C operation} \]
What happens when Data is put in?

We put in GO1.

1. Secretary (interface element) reports to director:
   "Somebody wants GO1!"

2. Director (CPU = Central Processing Unit = Microprocessor) asks his specialists:
   "Can we execute GO1?"

3. The specialists (EPROM = Programmable read-only memory) think and inform the director:
   "Yes we can!"

4. The director instructs the memory operating program (RAM = Random access memory):
   "Remember GO1!"

5. The memory reports to the director:
   "O.k., I have noted it down!"

6. Director instructs his press-speaker (output element):
   "Show them out there, that we are clear with GO1. We have everything understood and are ready for further inputs!"
What happens when Data is put in?

Data Input

Interface element
(secretary)

Central processing unit = Microprocessor
(Director)

Operating program = EPROMS (Specialists)

Memory = RAM

Output element
(press speaker)

Digital read-out

Data Input
The Block Format or Input Format

According to the key number (G-, M-functions) you have to put in the required information. The computer will ask these informations.

Example:

If you press INP after the G90 input, the indication jumps to the next block number.

Example:

You have entered the X,Y-values with GOO. After the registration of the Y-value the indication jumps to the next block number.

Why?

The computer knows that it can interpolate only in two planes. After input of X- and Y-values it sets the Z-value automatically to 0 (with incremental programming).

Example:

If you, however, have programmed the X-value with zero, the computer will ask for a Z-value.

Example:

With absolute programming mode the computer asks all three values X,Y,Z.

You have to tell the computer the plane from which it has to start the movements.
Indication on the Screen

Mode of operation absolute – incremental:

1. When switching on the CNC-operation the control is in incremental operating mode.

2. If you program G90 or G92 the screen shows the absolute operating mode.

3. If you program G25 or G27 the display disappears. The computer recognizes this only in the program run.

Mode of operation metric – inch:

According to the position of the option switch the metric or inch mode of operation will be indicated.

Metric 0.01 mm
Inch 0.001"

Vertical or horizontal axis system

These symbols indicate which axis system is in operation.
Input of program

Example

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (I)</th>
<th>Y (J)</th>
<th>Z</th>
<th>F (U)</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>00</td>
<td>3000</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>01</td>
<td>00</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2000</td>
</tr>
<tr>
<td>02</td>
<td>M30</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

1. Switch on main switch
Control lamp for current supply and lamp for mode of operation "hand-operation" are on.

2. Press key \(\text{H/C}\)
The control unit is switched over to CNC-mode of operation. On the digital read-out the lamp of address N is on. 00 (NOO) is shown.
The screen shows N 00.

3. Press key \(\text{INP}\)
With \(\text{INP}\) you instruct the computer to register NOO. The address letter jumps to G.

4. Put in G-information
00 shows on the digital read-out.

5. Press key \(\text{INP}\) Address indication jumps to X.
6. Put in X-value **3000**.

7. Press **INP**. Display jumps to Y.

8. Put in Y-value **0**.

9. Press **INP**. Display jumps to Z.

10. Put in Z-value **0**.

11. Press **INP**. Block NO0 is entered. Block indication jumps to NO1.

12. Enter block NO1 in the same way. Put in the Minus sign after the number value.

13. **M30** (program end)

   - Put in NO2
   - Display is at G.
   - Press key **M**, then the M-address is indicated.
   - Put in the figure value.
   - Press **INP**.

14. Press key **START**. Display jumps to NO0 (only if **M30** is programmed).

15. Press key **START**, the program runs.
Operating Elements – CNC

Program Input

Option Key Hand-Operation/CNC-Operation

By pressing key \( \text{H/C} \) the mode of operation changes from "hand-operation" to "CNC-operation".

The relative mode of operation is indicated by the lamps \( \Box \) (CNC-operation) or \( \circ \) (hand-operation).

To put in a program it has to be switched to CNC-operation.
In the CNC-mode of operation you cannot move the slides by hand anymore.
The Word Indication

The lamps and light bars of the word indication show you which data you can put in.

Digital read-out
The actual words are indicated by lamps.

```
N  G  X  Y  Z  F
O  O  D  J  K  L  T  M
```

Monitor
The actual words are indicated by a light bar.

```
CNC-OPERATION
N  G  X  Y  Z  F
00  
01  
```

Address indication – G, M function
If depends on G or M-functions which addresses and/or data are required?

E.g. M06
M06 requires a D,S,Z,T information.

Digital read-out
The X-indication is also valid for the D-value, the Y-indication for the S-value and the F-indication for the F-value if M06 was programmed.

```
N  G  X  Y  Z  F
O  O  D  J  K  L  T  M
```

Monitor
The address letter D,S,T are indicated.

```
CNC-OPERATION
N  G  X  Y  Z  F
00  
01  
02  
03 M06 D  S  T
```
The Indication of Addresses
D, J, K, L, T, M on the Screen

**G25/G27**
The address letter L is indicated.
(L = jump address, subroutine address)

**Format M06**
Addresses
- D (milling cutter radius)
- S (spindle speed)
- T (tool number)
are indicated.

**Format M99**
Addresses
- J (start of arc of circle)
- K (end of arc of circle)
are indicated.

**Attention:**
X,Y,F lamps are valid for various addresses.
The Figure Keys

You use the figure keys in order to enter the various values for address letters X,Y,Z,F,G,M,D,T,L,J,K. The entered values appear on the digital read-out and/or on the screen of the monitor.

The Minus-Sign Key

X,Y,Z values can have a minus or a plus sign.

Plus sign input for X, Y, Z:
Put in figures only.

Minus sign input

After input of figures, press key. The minus sign appears as a bar on the digital read-out.

Example:
X = -1400
Input: 1400-
The **INP** Key = Memory Key

**INP** = Abbreviation for Input

**INP** = Instruction to the computer to register the entered value.

---

**Example**

- Lamp X lights up.
- Enter value **2350**. The number appears for your information only, it is not in the computer yet.
- You press **INP**. By pressing this key, figures are registered; at the same time the number 2350 disappears and the light jumps to the next address letter.

---

**Note**

With **INP** you can also jump forward in the block.
The → Key

Instruction: to jump forward within one block

By pressing the key → the program will jump to the next word. The entered value of the next word will appear on the digital read-out.

(Permanent function when you keep on pressing the key)
The **FWD** Key

Instruction: to jump forward block-by-block

1. A given word is displayed. By pressing the **FWD** key the program jumps to the next block number.

2. If a block number is indicated: when pressing the **FWD** key the program jumps to the next block number.

3. If you keep the **FWD** key pressed down, the program will jump block-by-block to the program end.
The \textbf{REV} Key

Instruction: to jump back in program block-by-block

Function:

1. A given word is on the display. If you press key \textbf{REV} the program jumps to block number N.

2. If block number N is indicated and you press key \textbf{REV}, then the program will jump to the previous block number.

3. If you keep the \textbf{REV} key pressed the block number jumps back to NOO (permanent function).
"DEL" is the abbreviation of "delete", which means to cancel, to extinguish.

You can delete only the value of the address letter which is indicated. If you correct a X-value e.g., the address letter X has to be on the digital read-out.

Attention:

With DEL only the digital read-out is cancelled, not the value in the register. You must put in a new value and store it with INP.

Example: You want to change value X from 520 to 250.

1. Press DEL key, the value 520 will disappear.

2. Put in the correct value (250).

3. Press INP key, value X is registered; light jumps to the next address letter.
Input of M-Values

If you want to put in M-values: at first you have to select the M-key. The M-value is programmed in the G-column.

Digital read-out

Example

Monitor

Input: M30
Address G has to be shown
Press M
Put in 30
Press [INP] (register)

Attention:

+ M-values are not taken over by pressing [INP]

+ If you press [INP] after M30, the program jumps back to NOO.
Take-Over of registered Values into the following Blocks

By pressing [INP] the register takes over the previously entered value of the relative word column.

Example 1
- G-address is shown
- [INP]
- G-value flashes shortly and is registered
- Word indication jumps forward

Example 2
- You want to put in the value Z=0 in block NO3.
- You happen to see that the Z-value in block NO1 should be -1000 and correct the value.
- After correction you carry on with the Z-value input of block NO3.
- If you press [INP] the register takes over the previously entered Z-value, i.e. -1000.

Attention:
M-values and inputs are not taken over with [INP].
Inserting and Deleting of Blocks

\[ \sim + \text{INP} = \text{Inserting a block} \]
\[ \sim + \text{DEL} = \text{Deleting a block} \]

Remark 1:
First press key \( \sim \) and then key \( \text{INP} \) (keep \( \sim \) pressed).

Remark 2:
Permanent function when you carry on pressing (more than 0.6 sec.), i.e. you insert permanently empty lines with G21.

Example: Inserting \( \sim + \text{INP} \)

+ Digital read-out shows block NO2.

+ Press \( \sim + \text{INP} \)

+ In block NO2, G21 is automatically written.

+ The original block NO2 is automatically changed over to NO3 - also all subsequent blocks to the next block number.

+ In block NO2 you can program required instructions as you want.

Procedure
+ Delete G21
+ Put in wanted block

Example: Deleting \( \sim + \text{DEL} \)

+ Digital read-out shows NO2
+ Press \( \sim + \text{DEL} \)
+ NO2 is deleted
+ All subsequent blocks are backnumbered: NO3 - NO2, NO4 - NO3, etc.
Deleting of a registered Program

Possibility 1
Switch off main switch.

Possibility 2
Press emergency stop button.

Possibility 3
A certain block number is indicated (NOO, NO1, NO2 ...).

Procedure
First press key DEL then INF (DEL remains pressed).

The registered program is deleted. The digital read-out shows NOO.
The Program Sequence

1. Testrun
The program runs in the computer. There are no instructions given for slide movements.

2. Single-block operation
The program is worked off block by block. The slides move as programmed.

3. Automatic operation
The total program is worked off. Switching instructions are carried out.
1) Testrun

The program runs "in the mind". The instructions for slide movements are not given.

Purpose of the testrun:
- Block mistakes are shown.
- With absolute programming mistakes of the linear or circular interpolation are indicated (e.g. if you programmed movement in 3 planes simultaneously or you determined the target point of the quadrant incorrectly, etc.).

If you have programmed subroutines or jump instructions you can check the order of the instructions.

Activation of testrun:

1. CNC-operation

2. Indication has to be on N-address

3. Press M-key:
   the indicated block is worked off.

4. Press M-key:
   The following block is worked off.

etc.
2) Single block operation

In the testrun you do not see whether you run with e.g. GOO into the workpiece or whether the directions are correct. This you see in the single block- or in the automatic operation.

Example:

1. Block NO00

- Block indication is at NO00.

- Press key 1, then key START (key 1 has to remain pressed).

Block NO00 is worked off.

The screen shows "dwell in block NO01".

2. Block NO01

Press again 1 + START.

Block NO01 is worked off.

The screen shows "dwell in block NO02".

In this way the program can run in single block operation.
Single block operation (continued)

Various blocks in single block operation:

If you e.g. press keys $3 + \text{START}$, there will be 3 blocks worked off. You can work off up to 9 blocks in one go ($9 + \text{START}$).

Dwell in single block operation

Press $\text{FWD}$.
The slides stop.

If you press $\text{START}$, the program continues.

Interruption of program

Press $\text{REV}$.
The program jumps back to NO00.
3) Automatic operation

- Set block indication to NO00.

**Possibility 1**

Press [REV] key, until NO00 is indicated.

**Possibility 2**

Display shows any given block number. Press [INF + REV], indication jumps to NO00.

- Press key [START]. The program runs until a hold or until M30.

**To continue program after hold**

Press key [START].

**Program Hold**

- Programmed hold M00.

- In connection with M06, if under the address T (F) a number 1 to 499 is programmed (with inch operating mode 1 to 199). If under T=0 is programmed, there is no hold.
Interventions during Program Flow

1. Program stop
2. Program interruption

1. Program stop

Press keys [INF + REV]. The program jumps back to NOO (start).

Pay attention:
If you press [START] key after [INF + REV], the program starts with NOO. Your tool is not in starting position! Collision!

New start: Measures
Position the tool in program start position.

Sonst Kollisionsgefahr und falscher Programmablauf
2. Program Interruption

The program is stopped.

To continue program:
Press key START.

Why program interruption?
You may e.g.
- change the feed
- take measurements
- switch over to hand operation and carry out a correction by hand
- correct program, etc.

Effectiveness of Corrections with Program Interruption

1. Corrections of feed:
   Feed corrections become effective in the interrupted block.

2. Corrections of G,M,X,Y,Z-values in the interrupted block are only effective in the following program run.

3. Corrections of G,M,X,Y,Z-values in subsequent blocks will be effective when the program is continued.
9. ALARM SIGNS

- Purpose of alarm signs 9.1
- Procedure in the computer when input is wrong 9.2
- Alarm survey, possible inputs 9.4
- Measure when alarm sign appears 9.5
- Alarm signs, details 9.7 – 9.15
A05: M30 instruction missing

With START the computer checks if M30 (program end) was programmed.

A06: M03 instruction missing

(M03 main spindle ON)

This alarm only appears if threading cycles are programmed.

Attention:
The main spindle switch has to be in CNC-position!

A08
A09
A10
A11
A12
A14

Compare tape operation

A13: Inch/mm or vertical/horizontal switch with full program memory

This alarm cannot be cancelled by [APP] + [REV]. You have to switch back into the original position. If you have put in a vertical mill program with switch position at horizontal mill, you have to enter the program new (with correct switch position).

A15: Wrong Y-value

For admissible data see chart.
**A16: Cutter radius data missing**

If a G72, G45, G46, G47, G48 instruction is called, there has to be programmed a MO6 information with cutter radius data (D) in one of the previous blocks. Without this information the computer cannot calculate the center point path.

**A17: Wrong subroutine**

If a subroutine is nested more than 5 times, an alarm is shown.

**A18: Movement of cutter radius compensation smaller 0**

*Example:* subtract cutter radius once

1. 

MO6/D500/S.../Z.../F...

G46

G00/X3000/Y=0/Z=0

Cutter moves 30 minus 5 = 25 mm

2. 

MO6/D500/S.../Z.../F...

G46

G00/X500/Y=0/Z=0

No movement

Cutter radius = traverse movement

3. 

MO6/D500/S.../Z.../F...

G46

G00/X300/Y=0/Z=0

Alarm

Movement X=300 is smaller than cutter radius. 300 minus 500 = -200.
Special case – Alarm A18 with pocketing

The first measure for the pocket has to be larger or equal.

| Cutter dia + 0,1 cutter dia. |

Example:

Cutter dia. 10 mm
Minimum measure for pocket

\[ d + 0,1 = 10 + 0,1 \times 10 = 10,1 \text{ mm} \]

Reason:

Finishing cut 2 x 0,1 R (radius) is fixed in cycle G72.
Alarm Signs

Purpose of alarm signs:

If you put in and store data which the computer does not know, if you forget something or program a wrong block, then the computer gives an alarm sign.

The alarm sign appears on the digital read-out in form of a certain alarm number, on the monitor you get a commentary too.
What happens when wrong data is put in — Alarm sign

We put in a X-value 50000, i.e. for the cross slide a traverse path of 500 mm.

1. The secretary (interface element) reports:
   "They want X = 50000!"

2. The director (central processing unit, microprocessor) asks his specialists:
   "Can we execute X = 50000?"

3. The specialists (operating program) answer:
   "No, Mister Director! X 50000 is too high!"

4. The director instructs his speaker (output element):
   "Tell them out there, we cannot do it! X 50000 is too high, put in alarm sign AO2!"
What happens when wrong Data is put in?

Data Input:

1. Interface element (Secretary)
2. Operating program = EPROM (specialists)
3. Central processing unit = Microprocessor (Director)
4. Data on digital read-out

Data on digital read-out

Output element (Press. Speaker)

Memory = RAM
Alarm Signs
(Survey)

AO0: Wrong G/M instruction
AO1: Wrong radius/M99
AO2: Wrong X-value
AO3: Wrong F-value
AO4: Wrong Z-value
AO5: M30 instruction missing
AO6: MO3 instruction missing
AO7: No significance

A08: Tape end with tape operation SAVE
A09: Program not found
A10: Writing protection active
A11: Loading mistake
A12: Checking mistake
A13: Inch/mm switching with full program memory
A14: Wrong mill head position/path unit with LOAD ⊥/M or ⊥/M
A15: Wrong Y-value
A16: Cutter radius data missing
A17: Wrong subroutine
A18: Movement cutter radius compensation smaller 0

Possible Inputs
(otherwise alarms possible)

<table>
<thead>
<tr>
<th>Metric Values</th>
<th>Fineness (mm)</th>
<th>Inch Values</th>
<th>Fineness (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>X₁</td>
<td>0-19999</td>
<td>1/100 mm</td>
<td>0-7999</td>
</tr>
<tr>
<td>X₂</td>
<td>0-9999</td>
<td>1/100 mm</td>
<td>0-3999</td>
</tr>
<tr>
<td>Y₁</td>
<td>0-9999</td>
<td>1/100 mm</td>
<td>0-3999</td>
</tr>
<tr>
<td>Y₂</td>
<td>0-19999</td>
<td>1/100 mm</td>
<td>0-7999</td>
</tr>
<tr>
<td>Z₁</td>
<td>0-19999</td>
<td>1/100 mm</td>
<td>0-7999</td>
</tr>
<tr>
<td>Radii</td>
<td>0-9999</td>
<td>1/100 mm</td>
<td>0-3999</td>
</tr>
<tr>
<td>D(X) milling cutter radius with M06</td>
<td>0-9999</td>
<td>1/100 mm</td>
<td>0-3999</td>
</tr>
<tr>
<td>F</td>
<td>2-499</td>
<td>mm/min</td>
<td>2-199</td>
</tr>
<tr>
<td>T(F) tool address M06</td>
<td>0-499</td>
<td>1</td>
<td>0-199</td>
</tr>
<tr>
<td>L(F) jump instruction G27</td>
<td>0-499</td>
<td>1</td>
<td>0-221</td>
</tr>
<tr>
<td>H(F) exit signs M26</td>
<td>0-299</td>
<td></td>
<td></td>
</tr>
<tr>
<td>J/K circular parameter</td>
<td>0-90</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Measures when Alarm appears

Alarm is on

INP + REV
Alarm indication disappears

DEL
Cancel wrong value

Put in correct value

INP
Store

Note:

- Alarm A13 can be cancelled only by operating the option switch metric/inch, horizontal/vertical.

- Alarm sign of tape operation please compare chapter tape operation.
**A00: Wrong G/M instruction**

Example: G12, M55

**A01: Wrong radius/M99**

Possibility 1: Radius larger than admissible values

Possibility 2: Wrong value for end coordinates PE of quarter arc

Example: incremental value programming

N.../G02/X1000/Y1500/

Coordinates X=1000/Y=1500 cannot be end coordinate of quarter arc.

Example: absolute value programming

<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>90</td>
<td></td>
<td></td>
<td>0</td>
<td>100</td>
</tr>
<tr>
<td>1</td>
<td>01</td>
<td>3000</td>
<td>2000</td>
<td>0</td>
<td>100</td>
</tr>
<tr>
<td>2</td>
<td>02</td>
<td>4000</td>
<td>1000</td>
<td>30</td>
<td>100</td>
</tr>
<tr>
<td>3</td>
<td>M30</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Alarm</td>
</tr>
</tbody>
</table>

- In block NO1 point P1 is programmed.
- In block NO2 the quarter arc is programmed (coordinate P2). The X,Y values are correct. The Z-value would mean a circular interpolation in space (helix). This the computer does not know.

The alarm sign in this example does not appear when the program is put in but is on during the test run, automatic or single block operation.

**Explanation:**

At program input the computer just checks the contents of one block, it does not check the Z-value of the previous block.
A02: Wrong X-value

Compare chart for admissible values.

A03: Wrong F-value

Compare chart for admissible values.

A04: Wrong Z-value

Possibility 1: Admissible Z-value surpassed (compare chart)

Possibility 2: Threedimensional movement with absolute value programming

This alarm appears only in the test run, single block or automatic operation because the mistake cannot be recognized at program input (computer does not check contents of previous blocks at program input).

Example:

<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>90</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>00</td>
<td>0</td>
<td>1500</td>
<td>3000</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>00</td>
<td>3000</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

Monitor shows: Wrong Z-value; the computer accepts the X,Y values since it can carry out this interpolation and indicates the value shown last as being wrong value.

Attention:

Maybe you wanted to program Z=0 and Y1500 instead of 0. The computer cannot know this. The computer indicates Z as wrong value since it does not know your thoughts.
Chapter 10
Casette Operation
RS-232 C Operation
Magnetic Tape Operation

The tape enables you to store programs and to feed them into the computer memory.

1. Storing on tape

To transmit from computer memory to tape: We call this mode of operation SAVE or CHECK.

2. From tape into computer

To transmit the program from tape into the computer memory: we call this mode of operation LOAD.

Some data

- Memory capacity per tape side: approx. 400 blocks.
- Operation time per tape side: approx. 90 sec.

Operation advice

1. Use only digital cassettes
2. Erase new cassettes completely (see page 7.23). The test impulse from the final control of the producer can cause Alarm A1 or A12.
3. Main drive motor must not run during LOAD, CHECK, SAVE and ERASE operation.
4. Do not put down tape near main motor.
Magnetic Tape Operation

Transmission of a program from machine memory to magnetic tape

Mode of operation

SAVE = transmit from machine memory to magnetic tape
CHECK = control of transmitted (loaded) program

1. Press key \( \rightarrow \) until word indication G lights up. Press key \( \text{DEL} \).
   The indicated value disappears from the digital read-out.

   Press keys \( 6 \, 5 \, \text{INP}. \) On the read-out you see C indicated. \( C \, \_ \, \_ \, \_ \, \_ \) magnetic cassette tape operation.

3. Press key \( \text{FWD} \).
   On the read-out appears \( C \, F \, _ \, _ \).

4. Put in program number.
   You can put in figures \( 000 \) - \( 099 \)
   \( 00 \) - \( 09 \)
   \( 0 \) - \( 999 \)
   The sequence of the figures can be chosen as you like. Example for input of a program with number 76: Press keys \( 7 \, 6 \).

5. Press key \( \text{INP} \).
   The transmission / loading starts.

5.1. First free space on the tape is sought.
   If there are not data on the tape, it will advance approx. 4 seconds and rewind approx. 2 seconds.

Band without data on:

<table>
<thead>
<tr>
<th>4 sec. advance</th>
<th>2 sec. rewind</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tape begin</td>
<td>Tape end</td>
</tr>
</tbody>
</table>

Transmission SAVE
If there are already data/programs loaded on the tape, then the tape will advance to the end of the program which was loaded last. Then advance 4 seconds and rewind 2 seconds.

Tape with programs already loaded:

\[
\begin{array}{c}
\text{4 sec. advance} \\
\text{2 sec. rewind} \\
\text{2 sec.} \\
\text{2 sec.}
\end{array}
\]

Transmission SAVE

5.2. Transmission operation SAVE

The digital read-out indicates \[ \text{SAVE} \]
SA is the abbreviation for SAVE.
The program/data are "saved" from the machine memory - where they could be deleted - onto the tape.

5.3. At the end of the transmission operation the tape rewinds to the tape start.

5.4. Control operation CHECK

The digital read-out indicates \[ \text{CHECK} \]
The data in the machine memory are compared with the data loaded on the tape.
If you have already programs loaded on the tape, then the digital read-out will indicate these on the read-out whilst the tape advances. It will advance to the program loaded last and then the "CHECK" will be carried out.

CHECK of loaded program

\[
\begin{array}{c}
\text{Program 3} \quad \text{Program 2} \quad \text{Program 1}
\end{array}
\]

6. After CHECK the tape rewinds. The program is loaded on the tape.

Please never take out tape during operation!
1. Press key \( \rightarrow \) until word indication G lights up. If a figure of the G-function appears, press key DEL. Then indication on read-out disappears.

2. Put in G65. Press keys 065 INP. Read-out indicates C P

3. Press key INP. Read-out indicates C P

4. Put in number of program. E.g. for program number 76 you press keys 7 6. On read-out: C P 7 6

5. Press key INP.

5.1. The program number 7 6 is looked for. If you have other programs on the tape already, then these numbers appear on the digital read-out. E.g. C P 7 4 or C P 7 5

5.2. Loading: When the wanted program 76 is found, the loading operation starts. On the digital read-out you see C L O. LO is the abbreviation for "load".

5.3. After the loading is done, the tape rewinds. The read-out shows NOO. Program number 76 is stored in the machine computer.

6. If you press key START then the program starts operating.

---

Program 76  Program 75  Program 74  Tape begin
C L O  C P 7 5  C P 7 4
### From tape to machine

<table>
<thead>
<tr>
<th>LOAD</th>
<th>SAVE, CHECK</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Put in G 65</td>
<td>1. Put in G65</td>
</tr>
<tr>
<td>G</td>
<td>G</td>
</tr>
<tr>
<td>![65]</td>
<td>![65]</td>
</tr>
<tr>
<td>C</td>
<td>C</td>
</tr>
<tr>
<td>![..]</td>
<td>![..]</td>
</tr>
<tr>
<td>C</td>
<td>C</td>
</tr>
<tr>
<td>P</td>
<td>P</td>
</tr>
<tr>
<td>![..]</td>
<td>![..]</td>
</tr>
<tr>
<td>4. Put in program number</td>
<td>4. Put in program number</td>
</tr>
<tr>
<td>C</td>
<td>C</td>
</tr>
<tr>
<td>P</td>
<td>P</td>
</tr>
<tr>
<td>![..]</td>
<td>![..]</td>
</tr>
<tr>
<td>Program is sought and will be loaded in machine.</td>
<td>- Free space on tape is sought.</td>
</tr>
<tr>
<td>![C L O]</td>
<td>![C S A]</td>
</tr>
<tr>
<td>6. If program is loaded in machine, then read-out indicates:</td>
<td>- Machine program is transmitted/loaded on tape (SAVE)</td>
</tr>
<tr>
<td>![N 00]</td>
<td>![N 00]</td>
</tr>
<tr>
<td>Program can be started.</td>
<td>- Loaded program on tape is checked/compared with machine program.</td>
</tr>
<tr>
<td>![C C H]</td>
<td></td>
</tr>
</tbody>
</table>

### From machine to tape
Alarm Signs – Tape Operation
(Summary)

A08 - Tape end reached during loading of program from machine memory to tape (only with mode of operation SAVE)

A09 - Selected program cannot be found (mode of operation LOAD). Tape is full. M06 is not put in in selected program (mode of operation LOAD).

A10 - Writing protection active

A11 - Loading mistake

A12 - Checking mistake

General

When switching off machine (also when current breaks down) an interference pulse is put onto the tape. This interference pulse does not have any effect since the loading start only after 2 seconds of tape advance.

Thus:

Tape has to be rewind (automatically). Never take tape out during rewind operation.

---

Tape begin

First program starts after 2 seconds.

Empty space, interference pulses ineffective.
Alarm sign A08:

Only when using mode of operation "SAVE"!

**Reason**

Tape finish during loading (SAVE) from machine memory to tape.
(A08 only when using mode SAVE)
Alarm sign A08 appears on digital read-out.

**Measures**

- Press \[\text{FIN}\] and \[\text{REV}\]. Tape rewinds to tape begin.
- Digital read-out indicates NO0.
- Put in new tape and repeat loading operation.

**Attention:**

If you put in this tape and want to load the next finished program (transmit from tape to machine memory) A09 appears "No program end found!"
Alarm sign A09:

Only when using mode of operation "LOAD"!

A09 – Reason 1
Selected program not found.
If you call a non-existing program number when "loading" (from tape to machine memory), then alarm A09 appears.

Measures
- Press [INF] + [REV]
The tape rewinds. The digital read-out indicates after that NOO.
- Look for program on another tape (in case you are sure you put it in).

Example: You look on this tape for program no. 5

| 19 | 22 | 17 | Pr.Nr16 |

A09 – Reason 2
Selected program not fully on tape (MO6), since tape was finished when loading from machine memory to tape (already in mode of operation SAVE you had alarm A08).

Measures
- Press [INF] + [REV]
The tape rewinds, read-out indicates NOO.
- Look for program on other tape (in case you are sure that you put it in)

Example: You call on program no. 19

Program 19 does not have MO6, thus alarm A08 was indicated already during mode of operation SAVE.
A10 – Writing protection active:

Only when using mode of operation “SAVE” and “ERASE”!

If you remove the writing protection (i.e. the black caps) you cannot put any more data on this tape side.

If you put in such a tape side and you want to transmit a program from the machine memory to the tape, alarm sign A10 appears.

Measures:
- Press [INP + REV]
- Tape rewinds, put in other tape or mount writing protection again.

A11 – Load mistake:

Only when using mode of operation “LOAD”!

A11 – Reason 1
Motor is switched on or is being switched on during loading (tape-machine).
The program on the tape was not destroyed by switching on the motor.

Measures
- Switch off motor
- Press [INP + REV]
The tape rewinds, the read-out indicates NOO.
- Repeat loading operation.
- If you have All indicated also with the following loading operation, please see reason 2.

A11 – Reason 2
The program on the tape is destroyed. The reasons for it could be a mechanical fault on the tape, a power failure - or the machine was switched off when tape was not rewound.

Measures
- Transmit program to new tape.

Summary measures

<table>
<thead>
<tr>
<th>ALARM All</th>
</tr>
</thead>
<tbody>
<tr>
<td>Repeat loading</td>
</tr>
</tbody>
</table>

- No alarm All
- Reason was interference when loading
- Alarm All
- Reason was mistake on tape
A12 – Check mistake:

Only when using mode of operation "CHECK/SAVE"!

Possible reasons:
- Tape faulty
- Interference pulse: main motor switched on, short power failure, interference pulse from electrical conductor (lightning, switching on of soldering transformer ...)

The interference pulses can happen both when using mode of operations SAVE or CHECK.

Alarm sign A12 in mode of operation "SAVE" – Remedy

Store program under another number.

Explanation:
You cannot delete the false program just by its own. Thus you have to give to this program a new number, if you store in on the same tape. If you would use the same program number, then alarm A11 would appear when loading (tape - machine) since only the first one of two identical program numbers can be called on.

Measure:
- Put in $\text{INF} + \text{REV}$, tape rewinds, read-out shows NOO.
- Put in same program under a new number.
- If alarm A12 appears again, then tape is defective.

Interference during SAVE

Same program has to be put in under new program number.
Alarm sign A12 in mode of operation "CHECK"

During CHECK operation there may occur an interference impulse and alarm sign A12 will be indicated, without a defective tape being the reason.

Check:
- Press [INP] + [REV].
  Tape rewinds to begin, on read-out NOO.
- Load tape into machine memory. If there is no alarm A12 when loading, then the program is o.k.
- During loading A12 is indicated: the following is necessary - New tape, delete complete tape or put in program anew under another number.

Measures - Summary

\[
\begin{align*}
\text{Repeat loading} \\
\quad \text{No alarm A12:} \\
\quad \quad \text{Tape o.k.} \\
\quad \text{Alarm A12:} \\
\quad \quad \text{Tape defective} \\
\quad \quad \quad \text{- New tape} \\
\quad \quad \quad \text{- Delete tape} \\
\quad \quad \quad \text{- Put in program under another number.}
\end{align*}
\]
Mode of operation "ERASE"

Mode of operation "ERASE" (Erasing the tape)

1. Press key ➔ until word indication G lights up. If you see a figure of a G-function indicated on the digital read-out, then press [DEL].

2. Put in GU

   Press A ⌼ INP, on the display you see C

3. Press ➔ + [DEL] at the same time, on the display you see C E

   The tape is erased.
   After that the read-out shows NOO

Program Interruption during Tape Operation

Only when using mode of operation LOAD, CHECK, ERASE.

Program interruption

Press [INP] + [REV],
Tape rewinds to tape begin.

Why program interruption?

When using mode of operation LOAD:

If you find out that you called a non-existing program. If you press [INP] + [REV] the tape will not advance to the tape end but rewind immediately.

When using mode of operation CHECK:

If you do not want to wait for CHECK operation.

When using mode of operation ERASE:

It is enough that you erase about 10 seconds. When loading anew the tape machine will erase automatically all other remaining data.
When putting in the Tape, pay Attention:

1. Putting in with left spool full

- If you switch off the machine, the tape advances 1 second.

- If you switch on the machine, the motor rewinds the tape 2 seconds. So it is made certain that the tape is at the very beginning.

2. Putting in with right spool full

- If you put in the tape and program G65, then the tape rewinds to the beginning.

- If you put in the tape and not program G65, and switch on and off the machine, the following happens:

Switch on:

The tape rewinds 2 seconds.

Switch off:

The tape advances 1 second.

If you carry on like this, the tape moves further through the switching on and off and you get an interference pulse on the tape. A stored program will be registered.
RS-232 C is an international standardized Interface. It is an interface for information interchange. Via this Interface data can be transmitted to peripheric apparatus and vice-versa.

The data are transmitted via a cable. For the specific apparatus a cable has to be connected by an expert. The description how to connect cables are found in the wiring diagrams of the producers.

### Some Examples

**Connecting a paper tape puncher and paper tape reader**

The program of the Fl-CNC can be punched on a paper tape:

*Vice-versa:* From a paper tape the program can be transmitted to the Fl-CNC.

**Printing a program**

Via the RS-232 C Interface the program in the Fl-CNC can be printed on a list.

**Connection of computers**

Via RS-232 C computers and computer systems can be linked to the Fl-CNC. Programs can be transmitted to the Fl-CNC and vice-versa.

For computer connection a specific Software is necessary. The Software is an encoding information which "translates" the code of the computer to the code of the machine. This Software has to be written by an expert for the specific computer type.
Activating RS 232:

RS 232 is activated via G66. G66 does not enter the memory, it is a switching function.

Examples:

- Transmission from paper tape to memory of F1-CNC
  (With "request to send" signal)
  - Switch to CNC-mode (memory must be empty)
  - Insert paper tape
  - Start paper tape reader

1. Program G66

2. Press [INF]
   On the display appears [A]
   (A is the abbreviation for ASCII = American Standard Code for Information Interchange)

3. Press [INF]
   The display shows [A L O]
   (LO = LOAD)
   The program is transferred. At the end of the transfer the display shows [N 00]
1. Program G66
2. Press **INP** The display shows ![A](image)
3. Press **INP** The display shows ![A](image)
4. Start paper tape reader (transmission begins)

---

**Transmission from paper tape to F1-CNC (without "Request to send" signal)**

- Insert paper tape
- Switch to CNC-mode

---

**Transmission from F1-CNC to paper tape (with or without "Request to send" signal)**

- Switch to CNC-mode
- Insert paper tape
- Start paper tape puncher

1. Program G66
2. Press **INP** Display shows ![A](image)
3. Press **FWD** Display shows ![A](image)
   
   (SA = SAVE)

   The paper tape is punched.
<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Part Nr. | Part Name | Sheet Nr. | mm | inch |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Program Nr. | Name | Date | mm | inch |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
# Program Sheet EMCO F1 CNC

<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>F</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Part Nr. | Part Name | Sheet Nr. | mm | inch |
|----------|-----------|-----------|-----|------|
Program Nr. | Name | Date | mm | inch |
<table>
<thead>
<tr>
<th>N</th>
<th>G</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>F</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Part Nr. | Part Name | Sheet Nr. | mm | inch |
|---------|-----------|-----------|-----|------|
Program Nr. | Name | Date |   |      |
## PROGRAM SHEET EMCO F1 CNC

<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (J)</th>
<th>Y (K)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Measurement Units
- ☐ mm
- ☐ inch

### Sheet Details
- Part Nr.
- Part Name
- Sheet Nr.

### Program Details
- Program Nr.
- Name
- Date
<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L) (T) (H)</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Part Nr. | Part Name | Sheet Nr. | mm | inch
---|---|---|---|---
Program Nr. | Name | Date |
<table>
<thead>
<tr>
<th>N</th>
<th>G (M)</th>
<th>X (D)</th>
<th>Y (S)</th>
<th>Z</th>
<th>F (L)</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Part Nr. | Part Name | Sheet Nr. | mm | inch |
---------|-----------|-----------|-----|------|
|          |           |           |     |      |
|          |           |           |     |      |
|          |           |           |     |      |

Program Nr. | Name | Date |
-------------|------|------|
|             |      |      |
|             |      |      |
|             |      |      |
|             |      |      |
|             |      |      |
# Tool Data Sheet

<table>
<thead>
<tr>
<th>T1</th>
<th>T2</th>
<th>T3</th>
<th>T4</th>
<th>T5</th>
<th>T6</th>
<th>T7</th>
<th>T8</th>
</tr>
</thead>
<tbody>
<tr>
<td>d</td>
<td>d</td>
<td>d</td>
<td>d</td>
<td>d</td>
<td>d</td>
<td>d</td>
<td>d</td>
</tr>
</tbody>
</table>

\[ D = \frac{d}{2} \]

F

t

S

Hz

Hzk

<table>
<thead>
<tr>
<th>(d)</th>
<th>(mm)</th>
<th>Cutter dia.</th>
</tr>
</thead>
<tbody>
<tr>
<td>(D)</td>
<td>(mm)</td>
<td>Cutter radius</td>
</tr>
<tr>
<td>(F)</td>
<td>(mm/min)</td>
<td>Feed speed</td>
</tr>
<tr>
<td>(t)</td>
<td>(mm)</td>
<td>Max. milling depth</td>
</tr>
<tr>
<td>(S)</td>
<td>(U/min)</td>
<td>Spindle speed</td>
</tr>
<tr>
<td>(Hz)</td>
<td>(mm)</td>
<td>Difference measure</td>
</tr>
<tr>
<td>(Hzk)</td>
<td>(mm)</td>
<td>Corrected difference measure</td>
</tr>
</tbody>
</table>

### Zero-point of workpiece
- Start position
- Tool change position

### Zero-point offset (G92)
- \(X\) _______ mm
- \(Y\) _______ mm
- \(Z\) _______ mm

### Vertical axis system
- \(+Z\)
- \(+Y\)

### Horizontal axis system
- \(+Y\)
- \(+X\)
- \(+Z\)

- Drawing no.: 
- Denomination: 
- Workpiece material: 
- Program no.: 
- Name: 
- Date:
## Tool Data Sheet

<table>
<thead>
<tr>
<th>T1</th>
<th>T2</th>
<th>T3</th>
<th>T4</th>
<th>T5</th>
<th>T6</th>
<th>T7</th>
<th>T8</th>
</tr>
</thead>
</table>

\[
D = \frac{d}{2}
\]

- **d**: Cutter dia.
- **D**: Cutter radius
- **F**: Feed speed
- **t**: Max. milling depth
- **S**: Spindle speed
- **HZ**: Difference measure
- **HZK**: Corrected difference measure

### Vertical axis system

[Diagram of vertical axis system]

### Horizontal axis system

[Diagram of horizontal axis system]

**Zero-point offset (G92)**

- **X**: mm
- **Y**: mm
- **Z**: mm

**Zero-point of workpiece**

- Start position
- Tool change position

**Drawing no.:**

**Denomination:**

**Workpiece material:**

**Program no.:**

**Name:**

**Date:**
# Tool Data Sheet

<table>
<thead>
<tr>
<th>T1</th>
<th>T2</th>
<th>T3</th>
<th>T4</th>
<th>T5</th>
<th>T6</th>
<th>T7</th>
<th>T8</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Variables:**
- \( d \): Cutter dia.
- \( D = \frac{d}{2} \)
- \( F \): Feed speed
- \( t \): Max. milling depth
- \( S \): Spindle speed
- \( H_Z \): Difference measure
- \( H_{ZK} \): Corrected difference measure

**Axes:**
- Vertical axis system
- Horizontal axis system

**Zero-point of workpiece**
- Start position
- Tool change position

**Zero-point offset (G92):**
- \( X \) mm
- \( Y \) mm
- \( Z \) mm

**Additional:**
- Drawing no.:
- Denomination:
- Workpiece material:
- Program no.
- Name:
- Date: