Operating Instruction

EMCO COMPACT 5 PC

Edition 90-7

Ref. No. EN2 031

Operating Instruction
COMPACT 5 PC
90-7 EN2 031

Foreword

This software for the Compact 5 PC was developed by specialists, who have been working on software development for industrial systems for many years.

With the COMPACT 5 PC software you have not only an efficient CAD/CAM software, but you can also work like at a normal CNC machine and set up standard (DIN 66025 and ISO 1056) NC programs (software description chapter 6, Editor).

You can learn to operate a computer-controlled machine tool in a very short time by reading the respective manuals yourself, even if you do not have any previous programming knowledge. You should work through this software description step by step and **practise** all examples listed **directly on the PC**.

If you have any CAD knowledge, you can start with the specimen examples in chapter 5 by means of the good user guidance on the screen and the operating sheets (the last three pages of this manual).

TECHNICAL DOCUMENTATION
wishes you success

CONTENTS

A Description of machine 5

Accident prevention 5

Technical data machine 6

Technical data PC 7

Scope of supply 8

Setting-up 8

Connection of the machine 8

Description of the machine 9
- Main elements 9
- Key switch 9
- Emergency-off key 9
- Main spindle switch 9
- Adjusting the speed 10
- Main spindle speed display 11
- Travelling range of the slides 11
- Drive of slides 12
- Toolholder 13
- Tailstock 14
- Care of machine 14

Clamping devices for workpieces 15
- Summary 15
- 3-Jaw chuck ø 80 mm 16
- The collet chuck attachment 17
- The independent chuck 18
- The clamping plate ø 90 mm 18

Technological data 19
- Cutting speed 19
- Spindle speed 19
- Calculation of feed 19

Finding the cutting values 20
- Finding the spindle speed 20
- Finding the feed speed in mm/min 21

Service parts 23

B Software description **41**

1. Vocabulary explanations 41

2 Software installation, access into program **42**
2.1 Possible PC configuration 42
2.2 PC with a hard disk and 1 disk drive
installation, access 42

3. Explanations for the software **46**
3.1 The differences in the input keyboards 46
3.2 Important key functions 47
3.3 Coordinate system 48
3.4 Screen cursor 49
3.5 Menu structure - menu summary 50
3.6 Screen display 52
3.7 Selection / deselection of the menu points 52

4. Menu points **53**
4.1 Work piece **54**
4.2 Draw **55**
4.2.1 Cursor 56
4.2.2 Point 59
4.2.3 Line 61
4.2.4 Arc 64
4.2.5 Visible edges 68
4.2.6 Mirroring 68
4.2.7 Cleaning 68
4.2.8 Erasing the geometric element 69
4.2.9 Erasing all 69
4.2.10 Hotkeys - draw 70
Specimen example 1 71
Specimen example 2 72
Specimen example 3 73
4.3 Program **74**
4.3.1 Moving in the rapid 74
4.3.2 Moving with feed speed 74
4.3.3 Cycles 74
Roughing cycle 75
Back pocket cycle 77
Follow contour cycle 79
Cut-in cycle 81
Threading cycle 83
4.3.4 Machining possibilities: CAD/CAM - 90
4.3.5 Edit Tool change 91
4.3.6 Erase the last movement 91
4.3.7 Erasing the machining program 91
4.3.8 Entering the feed values 91
4.3.9 Change feed unit 91
4.310 Entering the finishing offsets 91

4.4 Display 92
4.4.1 Zoom all 92
4.4.2 Zoom window 92
4.4.3 Zoom work piece 93
4.4.4 Removing, inserting the head stock 93
4.4.5 Removing, inserting the work piece 93
4.4.6 Removing, inserting the tool 93
4.4.7 Removing, inserting the tool path 94
4.4.8 Inserting, removing the tail stock 94
4.4.9 Changing the simulation display 94
4.4.10 Shift Geometry 95
4.5 Set up 0,0 96
4.6 Manual 97
4.7 Machine 98
4.7.1 Automatic 98
4.7.2 Single Step 98
4.7.3 Fast run (Rapid) 98
4.7.4 Empty cut step 98
4.7.5 Repeating the program 99
4.7.6 Moving the tool to the cursor 99
4.7.7 Shifting the program 99
4.7.8 Altering the scale program 99
4.8 Archive 100
4.8.1 Storing the program 100
4.8.2 loading the program 100
4.8.3 Storing the geometry 101
4.8.4 loading the geometry 101
4.9 Print 101
4.10 End 101

5.Specimen examples 102

6. Edit 109

7.Notes for the software compilation 130

Work sheets: Menu summary, hotkeys (for taking out) 133

A DESCRIPTION OF MACHINE

Accident Prevention - Safety Instructions

- Note: the feed power of the slides is 1000 Newton (100 kp). If program is not correct, the workpiece could be pressed out of the clamping device when it is not clamped and supported correctly.
  Therefore use tailstock center for supporting the workpiece.

- Read instruction manual before working with the machine.

- Electrical connection: The electrical connections must be carried out professionally. A grounding receptacle must be available.

- Do not alter guards! Close belt guard before starting the machine. Never open belt guard while machine is running. Don't dismantle chip guard! Never work with dismanteled cover of E-control!

- Keep children and visitors away! The machine should be stored so that children and visitors not acquainted with the use of the machine cannot start it.

- Always wear safety goggles! Be also aware that some materials (for examp. brass) spray while being worked on. Therefore, it is important that all persons near the machine are protected.

- Wear proper apparel! Loose sleeves could get caught in chuck or workpiece.

- Keep work area clean! Cluttered areas and benches invite accidents.

- Remove adjusting keys and wrenches!
  Even when machine is not being used. The chuck keys should never be attached to the machine with chains or similar.

- Carry out measuring, adjusting, set-up, maintenance and resetting work only during standstill of the machine!

- Use chip hook for removing chips.

- Never touch running machine parts!
  Never try to stop workpiece or chuck with the hand.

- Do not surpass the clamping capacity of the lathe chuck and independent chuck!
  See maximum capacities.

- Be careful of extending chucks! Never reach over running (rotating) chucks.

- Switch machine off before servicing!
  Remove plug from socket.

- The cover of the E-control has to be removed by trained service people only.

- All electrical service and repair works (replacing fuses, boards etc.) have to be done by trained service people only.

- Use original spare parts, otherwise there is no title whatsoever to guarantee!

Technical Data Machine

Center height	65 mm	(2.559")
Distance between centers	310 mm	(12.205")
Swing over bed	130 mm	(5.118")
Swing over cross slide	60 mm	(2.362")
Travel of cross slide	50 mm	(1.969")
Approx. net weight	55 kg	(121 lb)

TAILSTOCK:

Center sleeve diameter	22 mm	(0.866")
Tailstock taper	MK1	(MT1)
Stroke of center sleeve	35 mm	(1.378")

TOOLHOLDER:

max. shaft height	12,5 mm	(0.492")

HEADSTOCK:

Hole through work spindle	16 mm	(0.630")
Spindle hole taper	MK2	(MT 2)
Speed range	200 - 2400 rev./min	
Spindle nose according to manufacturer's standard		

FEED MOTORS:

Step motor		5°, 50 Ncm	(0.37 ft.lbf)
Rapid feed	PC/AT, PC/386	700 mm/min	(27.559 in/min)
	PC/XT	350 mm/min	(13.780 in/min)
Feed power		approx. 1000 N	(225 lbf)

MAIN SPINDLE DRIVE:

Single-phase asynchronous motor

Electric supply	220 V AC / 50 Hz / 3,3 A
Power (60 % duty cycle)	800 W
Speed n	2650 min^{-1}

Technical modifications reserved!

Due to improvements and different modifications of the individual delivery countries, illustrations may diverge from the delivered machine.

Technical Data PC

Minimum configuration

IBM or compatible (PC/XT, PC/AT, PC/AT-386) with
- 1 hard disk 20 MB and
- 1 disk drive
- keyboard

Operating system	MS-DOS from version 3.1
Main storage	640 kB RAM
Graphic card	Hercules graphic card or EGA- colour graphic card
Screen	monochrome or colour display
Interface	parallel interface

Recommended configuration

IBM compatible PC/AT with
- 640 kB RAM
- 1 disk drive 1.2 MB
- 1 hard disk 44 MB
- 1 EGA - graphic card
- EGA - graphicable colour screen
- keyboard
- operating system MS-DOS version 3.3
- parallel interface
- co-processor

Note:
The software is valid for all 100 % IBM compatible PC's. Upon request Messrs. EMCO will indicate to you all computers on which this EMCO software was tested.

Modifications reserved!

Scope of Supply

Base machine
Operating tools
- Double open-jawed spanner SW 13x17
- Open-jawed spanner SW 8
- Ring spanner SW 13
- Hexagon socket screw key SW 8,5,4,3

Accessory:
Interface cable (2):
In addition a 25-pole interface cable is necessary for the operation of the machine to connect the machine to the PC. (Not included in the scope of supply!)

Dongle:
Is delivered with the EMCO-software. The dongle is a software-protection (protection against unauthorized operation).
If the dongle wouldn't be connected, following restrictions have to be expected:
- no possibilities to store or load programs
- editor locked
- no possibilities to print screen hard-copies
- no data-exchange to the machine

Setting-Up

Place the machine to a stable table.
Recommended height approx. 650 mm (25").

Connection of the Machine

Power supply
Plug in mains cable (1).
Electrical connections must be done professionally. A grounding receptacle must be available.

PC connection
Connect PC to the machine with the interface cable (2) via parallel interface (connection X7 at the rear side of the machine).

Description of the Machine

1 Machine bed
2 Chip container
3 Longitudinal slide
4 Cross slide
5 Headstock
6 Tailstock
7 Base for toolholder
8 Toolholder
9 Chip protection device
10 Stepping motor
11 Switch board

Key switch (1)

At position 1 of the key switch the machine is ready for operation (pilot lamp (2) lights up);
Main and feed motors are supplied with power.

Note:
To protect the machine against unauthorized starting, the key should always be taken off.

Emergency-off key with locking (3)

Upon actuation the power supply to main and feed motors is interrupted (pilot lamp (2) lights up).
Unlocking: Turn button in clockwise direction.

Main spindle switch (4)

The main spindle switch only remains in position 1 (main motor runs) if the key switch is in position 1, the emergency-off key is unlocked and if the gearbox cover is closed.

Note:
To avoid a too heavy warm-up of the stepping motors the machine should be switched off during a major standstill.

Adjusting the speed

Via the belt drive six different speeds can be adjusted on the main spindle:

Speed [rpm]	Belt position
200	BC1
330	BC2
550	BC3
950	AC1
1500	AC2
2400	AC3

Change of the V-belt

The V-belt from the motor pulley A to the transmission pulley B is always mounted.

Drive for range of revolutions AC1/AC2/AC3

From motor pulley A to main spindle pulley C.
The belt can be put on in 3 positions: AC1/AC2/AC3.
The pulley B runs idle.

Drive for range of revolutions BC1/BC2/BC3

Belt pulley B to belt pulley C (main spindle).
The belt can be put on in 3 positions: BC1/BC2/BC3.

Procedure

Open gearbox cover using the socket head screw (5).

Loosen hexagon nut (6) at the rear side (motor side) of the gearbox.

Lift motor.

Put belt to desired pulley position.

Press motor downward and tighten hexagon nut.

Close gearbox cover.

Main spindle speed display

Perforated disc and light barrier on main spindle

1. Function for all turning operations except when screw-cutting

Via hole series 1 and light barrier 1 the main spindle speed is indicated at the screen.

2. Function when screw-cutting

Via hole series 1 and light barrier 1 the main spindle speed is indicated at the screen.

The impulse from light barrier 2 via slot 2 is needed as an additional information for the angle position of the main spindle (synchronization of the main spindle with the slide feed).

Travelling range of the slides

The sketch shows the travelling range of the slides in the menue MANUAL (see software description) and without mounted clamping device.

Note:
Mind that the travelling range of the slides changes with the clamping device used and the way of clamping (three-jaw chuck, tailstock).

Drive of slides

Step motors:

Technical data:

Single step 5°
Torque 0,50 Nm (0.37 ft.lbf)

As the name says, a revolution of the motor is divided into steps.

Technical data:

- Traverse speed for longitudinal and cross slides:
  Rapid traverse speed
  700 mm/min (27.6 in/min) (PC/AT, PC/386)
  350 mm/min (13.8 in/min) (PC/XT)

  Programmable feed rates 2-499 mm/min resp. 0,002-0,499 mm/rev. (.08"-20 resp. .00008 - .02 in/rev.)
- Smallest possible traverse path: 0,0138 mm (.001")
- Traverse path longitudinal slide 300 mm (11.811")
- Traverse path cross slide 50 mm (1.969")
- Indication on screen in mm (in)
- Feed power on slides approx. 1000 N (225 lbf)

A revolution of the step motors is divided into 72 steps, i.e. one step = angle of 5°
(360° : 72 = 5°).

The limitation of the traverse paths (the Tack-Tack sound)

If you move the slides to the limit positions or against a stop, you will hear a tack-tack sound. The step motor receives impulses for further movement, but cannot move any further. That means overload on spindles, nuts and guideways of the slides.

Remedy: Stop machine

Ball screws - Preloaded nuts

Longitudinal and cross slides are driven via ball screws. The screws run play-free in the nuts (no backlash).

Reduction step motor - feed screws

Smallest slide movement (for longitudinal and cross slides)
When the step motor turns by 5° (with the smallest step) the slide will move 0,0138 mm (.001").

Traverse path indication on screen - slide movement

The traverse path will be indicated on the screen in 0,001 mm (.00004").

Steps (angle of step motors)	Traverse path (mm)	Traverse path (in)
1. Step (5°)	0,0138	.00054
2. Step (10°)	0,0277	.00109
3. Step (15°)	0,0416	.00164
4. Step (20°)	0,0555	.00219
5. Step (25°)	0,0694	.00273
6. Step (30°)	0,0833	.00328
7. Step (35°)	0,0972	.00383
8. Step (40°)	0,111	.00437
9. Step (45°)	0,125	.00492

The toolholder

The toolholder can be fixed in a front or back position on the cross slide.

Max. tool section: 12x12 mm (.47" x .47")

Mounting

1. Put base (1) on toolholder support (2) and fasten with socket head screw (3).
2. Loosen T-sliding block (4) with hexagon screw (5) and thread in the toolholder from the top.
3. Unscrew cylinder screw (6), turn the milled nut (7) in and out respectively until the main cutting edge of the clamped cutter is exactly at centre height. Turn in socket head screw (6), thus the milled nut is protected against twisting.
4. Clamp toolholder with hexagon screw (5) (in clockwise direction).

Positions of toolholder

The toolholder can be clamped in front position and in back position.

Front position

Outside diameter ⌀ 0 bis ⌀ 90 mm (0-3,543")

Interior diameter ⌀ 14 bis ⌀ 100 mm (0.551" - 3.937")

Back position

Outside diameter ⌀ 25 to ⌀ 130 mm (0.984"-5.118")

Interior diameter ⌀ 40 to ⌀ 130 mm (1.575" - 5.118")

Clamp the toolholder in the front position for programming exercises.

The tailstock

The tailstock serves to support the workpiece by using a center - as well as for drilling / centering.

Drilling operation

Drills up to ⌀ 8 mm (0.314") to be mounted in drill chuck. Drills of more than 8 mm need a MT1 so that they can be set in directly into the tailstock barel.
Feed via handwheel and tailstock sleeve.

Tailstock accessories

- Plug-in pivot: for mounting clamping devices (e.g. chuck)
- Lathe centre

Regular care of machine

Clean slideways and oil them with non-rusting slide oil. Main spindle bearing, recirculating ball screws are maintenance-free.

Clamping Devices for Workpieces (Summary)

Collet chuck attachment

Clamping capacity 1,5 - 14 mm in connection with collets type ESX 25.

Round workpieces can be clamped with highest round-run accuracy using the collets. Collets leave no clamping marks on the workpiece.

4-jaw independent chuck 90 mm diameter

With the 4-jaw independent chuck, workpieces can be clamped centrically and excentrically. Each jaw can be individually adjusted and reversed.

3-jaw chuck, 80 mm diameter

The 3-jaw chuck serves for centricalls clamping round, hexagon or twelve-sided workpieces.

Clamping plate, 90 mm diameter

For clamping large-dimensioned workpieces that cannot be clamped with the 3-jaw or 4-jaw independent chuck. The workpiece is clamped by using the clamping shoes.

3-Jaw chuck ⌀ 80 mm

For centrically clamping of round, hexagon and 12-sided workpieces. Square workpieces cannot be clamped centrically with the 3-jaw chuck.

Clamping capacity

If the clamping capacity would be exceeded, a tooth-breaking on the clamping jaw is possible. Safety clamping is not guaranteed.

	d_5 min	d_6 max	d_7 min	d_8 max	d_9 max	d_{10} min	d_{11} min	d_{12} max	d_{13} max
mm	1	36	21	80	104	5	29	84	104
inch	0.04"	1.4"	0.8"	3.15"	4.1"	0.2"	1.15"	3.3"	4.1"

Mounting

Spindle nose and chuck bore must be dust-free. Mount the 3-jaw chuck onto the spindle nose with the 3 screws (M5x40, DIN 912).

Do not use longer screws - this would prevent the correct contact with the spindle nose.

Do not use shorter screws - the screws could break or tear out.

Note:

A set of chucks graded outwards and a set graded inwards are included in the delivery.

The respective chuck number must be in accordance with the actual groove number.

Safety tip

Never exceed the maximum clamping capacity of the chuck.

This could cause the chuck teeth to break - the jaws and the workpiece would be thrown out and could cause severe injuries.

Working tips - parting-off

Clamp workpiece with as minimum overhang as possible, so that it is not bent by the pressure of the tool.

The parting-off tool must be clamped at exact center height and at a right angle to the turning axis.

Supporting long workpieces

Long workpieces bend through the pressure of the tool and must be supported by the tailstock center.

Employment of the rolling center punch

With spindle-speeds more than 550 rev./min the rolling center punch should be used.

Soft jaws for the chuck

The steps of the soft jaws must be turned by the user. When turning these steps, clamp a round workpiece to fix the jaws.

Procedure:

Clamp a round workpiece and turn the step.

Safety tip:

The overhang of the jaws may never be more than 12 mm (1/2"), otherwise the teeth might break. If the teeth break, the jaw could be thrown out and cause severe injuries.

The steps must be turned so deep that the workpiece is clamped securely.

The collet chuck attachment

Clamping capacity 1,5 - 14 mm (1/16" - 35/64") using the collets type ESX 25.
Round workpiece can be clamped with highest round-run accuracy using the collets. Collets leave no clamping marks on the workpiece.

Mounting
Mount collet holder (1) onto the spindle nose with the 3 hexagon screws (2).

Clamping the workpiece
Insert collet, loosen clamping nut (3), insert workpiece and re-tighten clamping nut with the socket head key.
Tightening is done clockwise.

Mounting of the collet chucks:
- Insert collet chuck (4) laterally into the clamping nut (3) so that the eccentric ring (5) engages in the groove of the collet chuck.
- Screw collet chuck with clamping nut onto collet chuck holder (1).

Taking-out of the collet chuck:
Untighten clamping nut.
Via the eccentric ring in the clamping nut the collet chuck is pressed out when screwing off the clamping nut.

Care/Service
Clean and oil collet holder before and after use - chips and dirt could damage the clamping taper and cone and influence the precision.

Clamping capacities

Nominal diameter	Clamping area in mm	Clamping area in inch
2,0	1,5 - 2,0	1/16 - 5/64
2,5	2,0 - 2,5	3/32
3,0	2,5 - 3,0	7/64
4,0	3,0 - 4,0	1/8 - 9/64 - 5/32
5,0	4,0 - 5,0	11/64 - 3/16
6,0	5,0 - 6,0	13/64 - 7/32 - 15/64
7,0	6,0 - 7,0	1/4 - 17/64
8,0	7,0 - 8,0	9/32 - 19/64 - 5/16
9,0	8,0 - 9,0	21/64 - 11/32
10,0	9,0 - 10,0	23/64 - 3/8 - 25/64
11,0	10,0 - 11,0	13/32 - 27/64
12,0	11,0 - 12,0	7/16 - 29/64 - 15/32
13,0	12,0 - 13,0	31/64 - 1/2
14,0	13,0 - 14,0	33/64 - 17/32 - 35/64

The independent chuck

For clamping round, square, rectangular and uneven shaped workpieces.
Each jaw can be adjusted individually.
Workpieces can be clamped centrically and excentrically.

Mounting
Mount the independent chuck to the spindle nose with the 4 screws (M5x25, DIN 912).

Clamping capacities

	d_5 min.	d_6 max.	d_7 min.	d_8 max.	d_9 max.	d_{10} min.	d_{11} min.	d_{12} max.	d_{13} max.
mm	1	42	21	86	110	9	29	90	110
inch	0.04"	1.6"	0.8"	3.4"	4.3"	0.35"	1.15"	3.54"	4.3"

The clamping plate 90 mm diameter

Clamping capacity using the small T-nut screws: up to 13 mm (1/2").

Clamping capacity using the big T-nut screws: up to 33 mm (1/3").

Often uneven or irregular shaped workpieces cannot be clamped with the 3-jaw chuck or the 4-jaw independent chuck.

Using the T-nut screws and the clamping shoes, uneven or large-dimensioned workpieces can be clamped. The rings turned into the clamping plate serve as an aid for centrical clamping.

Mounting
Mount the clamping plate onto the spindle nose with the 4 socket head screws M5x20.

Safety tip:
Uneven workpieces often cause unbalanced round-run. Therefore always work with low spindle speeds. Be careful of extending parts.

Technological Data

1. Cutting speed (V_S)

$$V_s\,(m/min) = \frac{d(mm) \times \pi \times S(rev./min)}{1000}$$

V_S = Cutting speed
d = Dia. of workpiece
S = Speed of main spindle

The max. acceptable cutting speed depends on:

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

- Material of tool:
  Carbide tools allow for a higher cutting speed than HSS tools.

- Feed:
  The larger the feed the lower the cutting speed.

- Depth of cut:
  The larger the depth of cut the smaller the cutting speed.

Cutting speed for programming exercises on the COMPACT 5 PC

Workpiece material: automatic aluminium
Tool: carbide tips
Cutting speed for turning: 150-200 m/min
Cutting speed for parting off: 60-80 m/min
Feed size for turning: 0,02 - 0,1 mm/rev.
Feed size for parting off: 0,01 - 0,02 mm/rev.

2. Spindle speed (S)

The cutting speed and the workpiece dia. enable you to calculate the speed of the main spindle.

$$S\,(rev./min) = \frac{Vs\,(mm/min) \times 1000}{d\,(mm) \times \pi}$$

3. Calculation of feed

On the COMPACT 5 PC you program the feed in mm/min

Conversion:

F (mm/min) = S (rev./min) x F (mm/rev.)

F (mm/min) = Feed in mm per minute
S = Speed of main spindle
F (mm/rev.) = Feed in mm per revolution

The charts on the following page save the calculation work.

Finding the Cutting Values

1. Finding the spindle speed (r.p.m)

You know

- Diameter of workpiece
- Suggested cutting speed

From the chart you can select the spindle speed in rpm.

Example:

Diameter of workpiece: 40 mm
Cutting speed: 150 m/min
Therefore: 1200 rpm

2. Finding the feed speed in mm/min

You know

- Diameter of workpiece
- Feed size in rpm

From the chart you select the feed in mm/min

Example:

Spindle speed: 1200 rpm
Feed: 0,06 mm/rev.
Results in feed speed: 70 mm/min

emco

Ersatzteilliste COMPACT 5 PC

Service Parts for COMPACT 5 PC

Pièces de service pour COMPACT 5 PC

Ausgabe 90/04

Edition 90/04

Edition 90/04

Pos.	Ref. No.	DIN		Benennung	Description	Designation
				Bedienungswerkzeuge	Service tools	Outils
	ZWZ 11 0300	SW3 DIN 911		6kt. Schraubendreher	Hexagonal key	Clé à six pans
	ZWZ 11 0400	SW4 DIN 911		6kt. Schraubendreher	Hexagonal key	Clé à six pans
	ZWZ 11 0500	SW5 DIN 911		6kt. Schraubendreher	Hexagonal key	Clé à six pans
	ZWZ 11 0800	SW8 DIN 911		6kt. Schraubendreher	Hexagonal key	Clé à six pans
	ZWZ 94 0800	SW8 DIN 894		Einmaulschlüssel	Spanner	Clé deservice
	ZWZ 95 1713	17x13 DIN 895		Doppelmaulschlüssel	Double ended spanner	Clé plate double
	C6Z 180 040	SW13		Ringschlüssel	Ring spanner	Clé polygonale

Pos.	Ref. No.	DIN		Benennung	Description	Designation
1	A6A o1o o1o			Bett	Bed	Banc
2	A5A o1o o2o			Spindelstock	Headstock	Poupée fixe
3	A5A o1o o3o			Reitstockgehäuse	Tailstock housing	Corps de la poupée
5	ZRG 28 oo8o	B8 DIN127		Federring	Spring washer	Rondelle ressort
8	ZSR 12 o845	M8x45 DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
1o	A6A ooo o6o			Abdeckung	Cover	Couvercle
11	ZSR 63 o4o8	M4x8 DIN963-4.8		Senkschraube	Countersunk screw	Vis tête fraise
12	A6A ooc o4o			Lagerbock 2	Bearing block	Palier de la roulement
13	ZSR 12 o53o	M5x3o DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux

36 34 35 37 38
32 27 26 24
25 23 22 21
20
33
19
EMCO
COMPACT 5 PC
5
6
7
2
3
4
16
12
14
15
28 22 31 29 30
17
18
8 1 9 10 11 12 13 14 15

Pos.	Ref. No.	DIN		Benennung	Description	Designation
1	A6A ooo o2o			Riemenscheibe	Pulley	Poulie
2	A5A ooo o3o			Motorriemenscheibe	Motor pulley	Poulie de moteur
3	ZSB 22 o53o	B5,3 DIN9o21		Scheibe	Washer	Rondelle
4	ZSR 84 o512	M5x12 DIN84-4.8		Zylinderschraube	Flat head screw	Vis à tête cylindrique
5	A5A o6o ooo			Vorgelegeriemenscheibe	Countershaft pulley	Poulie
6	ZSB 1o 2181	PS12x18x1,2		Stützscheibe	Supporting ring	Rondelle
7	ZSB 99 o9oo	9 DIN6799		Sicherungsscheibe	Retaining washer	Poulie de retenue
8	ZRG 71 2412	24x1.2 DIN471		Sicherungsring	Retaining ring	Anneau de retenue
9	A6A ooo 11o			Teilscheibe 1oo	Dividing washer 1oo	Disque diviseur 1oo
1o	ZSR 84 o512	M5x12 DIN84-4.8		Zylinderschraube	Flat head screw	Vis à tête cylindrique
11	A6A ooo 24o			Hülse	Spacer	Douille d'écartement
12	ZLG 6o o6o2	6oo6-2Z		Rillenkugellager	Ball bearing	Roulement à billes
13	ZSB o2 6oo6	6oo6/2K		Ausgleichscheibe	Compensating washer	Rondelle de compensation
14	ZSB 1o 5553	SS45x55x3		Stützscheibe	Supporting ring	Rondelle
15	ZRG 72 552o	B55x2 DIN472		Sicherungsring	Retaining ring	Anneau de retenue
16	A5A ooo o1o			Hauptspindel	Main spindle	Broche principale
17	ZRM 4o 6335	6x335		Keilriemen	V-belt	Courroie trapézoidale
18	ZRM 4o 645o	6x45o		Keilriemen	V-belt	Courroie trapézoidale
19	A6B ooo 17o			Frontschild	Front plate	Plaque frontale
2o	ZSR 33 o612	M6x12 DIN933-5.6		Sechskantschraube	Hexagon head screw	Vis hexagonale
21	ZMU 34 o8oo	M8 DIN934-6		Sechskantmutter	Hexagonal nut	Ecrou 6 pans
22	ZSB 21 o84o	A8,4 DIN9o21		Scheibe	Washer	Rondelle
23	A5A ooo 1oo			Lagerbolzen	Bearing shaft	Axe palier
24	A5A 13o ooo			Trägerplatte	Carrier plate	Plaque support seule
25	A6A o3o ooo			Spindelstockabdeckung	Headstock cover	Couvercle de la poupée fixe
26	ZSB 22 o53o	B5,3 DIN9o21		Scheibe	Washer	Rondelle
27	ZSR 11 o512	M5x12 DIN6912-6.9		Zylinderschraube	Socket head screw	Vis à 6 pans creux
28	ZMU 8o o8oo	NM8 DIN98o-8		Sicherungsmutter	Securing nut	Ecrou de sûreté
29	A6A 1o6 oo1			Lichtschranke	Light barrier	Barrière lumineux
3o	ZSR G3 3513	B3.5x13 DIN7981		Blechschraube	Sheet metal screw	Vis en tôle
31	A6A ooo 28o			Abstimmblech	Compensating sheet	Tôle de compensation
32	ZSB 12 12o3	PS 12x18xo,3		Paßscheibe	Shim ring	Rondelle
33	ZMO 17 1224		220 V	Motor	Motor	Moteur
	ZMO 19 1114		115 V	Motor	Motor	Moteur
	ZMO 17 1104		1oo V	Motor	Motor	Moteur
34	ZEL 21 2034	TS 231-2oY		Endschalter	Limit switch	Interrupteur de fin
35	ZEE 47 0010	ZV1 H231-11Y		Rollschwenkhebel	Control lever	Levier de commande
36	ZSR 79 0425	M4x25 DIN7991-8.8		Senkschraube	Flat head screw	Vis à tête conique
37	ZRG 26 0040	A4 DIN127-A2E		Federring	Split washer	Rondelle-ressort

6
24
33
10
9
11
32
24
4
2
5
7
29
8
3
1
16
17
18
19
20
18
21
23
24
22
31
26
30
29
27
28
12
13
14
15
25
26
27
28

Pos.	Ref. No.	DIN		Benennung	Description	Designation
1	A6A o2o o1o			Schlitten	Slide	Chariot
2	A6A o2o o2o			Querschlitten	Cross slide	Chariot transversal
3	A6A o2o o6o			Abdeckblech	Cover sheet	Couvercle
4	A6A o2o o7o			Einstelleiste	Gib	Lardon
5	A6A 021 ooo			Gruppe Querspindel	Cross slide spindle c.	Ens. broche transversale
6	ZSR 12 o5o8	M5x8 DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
7	ZSR 12 o51o	M5x1o DIN 912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
8	ZSR 84 o5o6	M5x6 DIN 84-4.8		Zylinderschraube	Socket head screw	Vis 6 pans creux
9	A6A o2o o8o			Gewindestift	Set screw	Vis pointeau
1o	ZST 13 o6o6	M6x6 DIN913-45H		Gewindestift	Set screw	Vis pointeau
11	ZMU 34 o4oo	M4 DIN934-5		Sechskantmutter	Hexagonal nut	Ecrou 6 pans
12	A6A 160 ooo			Gruppe Längsspindel	Lead screw complete	Ens. vis-mère
13	A6A o4o ooo			Abstreifblech	Wiper sheet	Tôle de racleur postérieur
14	ZSR 12 o4o8	M4x8 DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
15	ZSR 12 o516	M5x16 DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
16	A6A ooo 22o			Abstreiffilz 1	Felt wiper 1	Racleur en feutre post. 1
17	A6A ooo 2oo			Abstreifblech 1	Wiper plate 1	Plaquette de racleur post.1
18	ZSR 84 o3o8	M3x8 DIN84-4.8		Zylinderschraube	Flat head screw	Vis à tête cylindrique
19	A6A ooo 23o			Abstreiffilz 2	Felt wiper 2	Racleur en feutre post. 1
2o	A6A ooo 21o			Abstreifblech 2	Wiper plate 2	Plaquette de racleur post.2
21	A5A ooo 13o			Bettleiste kurz	Keep plate short	Lardon de chariot court
22	A6A ooo o5o			Bettleiste lang	Keep plate long	Lardon de chariot long
23	ZSB 25 o53o	B5,3 DIN125		Scheibe	Washer	Rondelle
24	ZSR 12 o512	M5x12 DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
25	A6A ooo o7o			Motorplatte 1	Motor plate 1	Plaque de moteur 1
26	ZSR 33 o51o	M5x1o DIN933-5.6		Sechskantschraube	Hexagon head screw	Vis hexagonale
27	ZSR 33 o4o8	M4x8 DIN933-5.6		Sechskantschraube	Hexagon head screw	Vis hexagonale
28	A6A 103 001			Schrittmotor X,Z	Step motor X,Z	Moteur pas à pas X,Z
	ZMO 78 0010			Schrittmotor X,Z ohne Kabel ohne Kühlkörper	Step motor X,Z without cable without dissipator	Moteur pas à pas X,Z sans câble sans refroidisseur
29	ZRM 73 48o5			Zahnflachriemen	Timing belt	Courroie crantée
3o	ZSR 12 o535	M5x35 DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
31	A6A ooo o8o			Motorplatte 2	Motor plate 2	Plaque de moteur 2
32	A6A ooo 26o			Anschlag	Stop	Butée
33	A6A ooo 14o			Stahlhalterauflage	Toolpost support	Support de tourelle

Pos.	Ref. No.	DIN		Benennung	Description	Designation
1	A6A o6o oo2			Spänetasse	Chip tray	Bac à copeaux
2	ZSR G1 3595	B3,5x9,5 DIN7981		Blechschraube	Sheet metal screw	Vis en tôle
3	A6A o5o ooo			Deckel	Cover	Couvercle
4	ZRG 71 o6o7	6xo,7 DIN471		Sicherungsring	Retaining ring	Anneau de retenue
5	ZSB 25 o64o	B6,4 DIN125		Scheibe	Washer	Rondelle
6	ZSR 12 o62o	M6x2o DIN912		Zylinderschraube	Socket head screw	Vis 6 pans creux
7	ZSB 21 o64o	A6,4 DIN9o21		Scheibe	Washer	Rondelle
8	ZGU 77 o621	17oo mm		Kantenschutzprofil	Protective profile	Perfil protective
9	ZMU 34 o6oo	M6 DIN934-6		Sechskantmutter	Hexagonal nut	Ecrou 6 pans
1o	ZSR 12 o616	M6x16 DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
11	ZEE 25 1o1o			Kabelklipp	Clip for cable	Pince pour câble
12	ZEL 15 ooo8			Tülle	Ring	Bague
13	ZMU 34 o8oo	M8 DIN934-6		Sechskantmutter	Hexagonal nut	Ecrou 6 pans
14	ZSB 21 o84o	A8,4 DIN9o21		Scheibe	Washer	Rondelle
15	ZSR o3 o82o	M8x2o DIN6o3-4.6		Flachrundschraube	Square neck bolt	Collet carré
16	A6D 2oo o5o		5o Hz	Drehzahlschild	Speed plate	Plaquette de vitesses
	A6C 2oo o5o		6o Hz	Drehzahlschild	Speed plate	Plaquette de vitesses

Pos.	Ref. No.	DIN		Benennung	Description	Designation
				Reitstock	Tailstock	Poupée
1	A5A 121 000			Pinole metr.	Barret metr.	Canon metr.
	A5B 121 000			Pinole zöllig	Barret inch	Canon en pouces
2	A5A 120 020			Lagerbüchse	Bearing bush	Coussinet
3	A5A 120 030			Triebschraube	Lead screw	Broche
4	A5A 022 000			Handrad	Hand wheel	Volant
5	A5A 021 050			Kegelgriff	Handle	Bouton cylindrique
6	ZSR 12 0540	M5x40 DIN 912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
7	ZMU 34 0500	M5 DIN 934-5		Sechskantmutter	Hexagonal nut	Ecrou 6 pans
8	ZMU 17 0800	M8 DIN 917-5		Hutmutter	Cap nut	Ecrou borgne
9	B2A 040 050			Knebelgriff	Locking handle	Vis à poignée
10	B2A 071 020			Scheibe	Washer	Rondelle
11	A6A 010 020			Hülse	Spacer	Douille d'ecartement
12	ZGF 51 1406	14xM6x40 GN/100		Klemmhebel	Clamping lever	Levier de serrage
13	ZSR 31 0635	M6x35 DIN 931-5.6		Zylinderschraube	Socket head screw	Vis 6 pans creux
14	ZST 14 0620	M6x20 DIN 914-45H		Gewindestift	Set screw	Vis pointeau
15	ZST 15 0601	M6x20 DIN 915 ORIL		Gewindestift	Set screw	Vis pointeau
16	A3A 000 040			Klemmplatte	Clamping plate	Plaque de blocage

Pos.	Ref. No.	DIN		Benennung	Description	Designation
	A6A 14o oo1			Gr. Späneschutz	Chip guard compl.	Ens. pare-copeaux
1	A6A 14o o11			Frontschutz	Front guard	Pare-copeaux
2	A6A 14o o21			Deckel	Cover	Couvercle
3	A6A 14o o31			Scharnier	Frame joint	Charnière
4	ZSR M3 o41o	M4x1o DIN 7985-4.8		Linsenschraube	Filister head screw	Vis à tête lentiforme
5	ZSR M3 o412	M4x12 DIN 7985-4.8		Linsenschraube	Filister head screw	Vis à tête lentiforme
6	ZSB 21 o43o	A4,3 DIN 9o21		Scheibe	Washer	Rondelle
7	A6A 14o o4o			Anschlag	Stop	Butée
8	A6A 14o o5o			Leiste	Gib	Barre

Pos.	Ref. No.	DIN		Benennung	Description	Designation
1-5	584 180			Schnellwechselstahl-halter	Quick-change tool-post	Tourelle porte-outils à changement rapide
2	ZSR 31 0860	M8x60 DIN931-5.6		Sechskantschraube	Hexagon head screw	Vis hexagonale
3	C6Z 180 020			T-Nutenstein	T-nut	Boulon en T
4	C6Z 180 030			Zwischenstück	Intermediate piece	Pièce d'écartement
5	ZSB 25 0840	B8,4 DIN125		Scheibe	Washer	Rondelle
6	ZSR 12 1050	M10x50 DIN6912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
7	ZSB 25 1050	B10,5 DIN125		Scheibe	Washer	Rondelle
8-12	584 181			Drehstahlhalter kompl.	Tool holder compl.	Ens. porte-outil
9	C6Z 181 020			Stellknopf	Adjusting nut	Ecrou de reglage
10	ZSR 12 0520	M5x20 DIN 912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
11	ZSR 13 0516	M5x16 DIN912-10.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
12	ZST 13 0525	M5x25 DIN913-45H		Gewindestift	Set crew	Vis pointeau
	584 280			Abstechhalter kompl.	Porting tool holder compl.	Ens. appui de outil à tronconner
14	C6Z 280 020			Spannplatte	Clamping plate	Plaquette de serrage
15	ZSR 12 0512	M5x12 DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux

e1 ..... 4A ..... ZEE 75 0011
e2 ....., 4A ..... ZEE 75 0011
e3 ..... 6,3A ..... ZEE 75 0013
e4 (220 V) ..... 2,5A ..... ZEE 75 0012
(115 V) ..... 4A ..... ZEE 75 0011
e6 ..... 1A ..... ZEE 75 0014
e7 ..... 16A ..... ZEE 70 2016
e8 (220-240 V)..... 8A ..... ZEE 75 0030
(100-120 V).... 10A ..... ZEE 75 0035

Pos.	Ref. No.	DIN		Benennung	Description	Designation
1	ZEE 53 7110		110 V	Signalleuchte	Signal lamp	Lampe de signalisation
	ZEE 53 7220		220 V	Signalleuchte	Signal lamp	Lampe de signalisation
2	ZEL 21 0014			Hauptschalter mit 2 Schlüssel	Main switch with 2 keys	Interrupteur général avec 2 clé
3	ZEL 22 0004	Bx9	100 V	Nockenschalter	Cam controller	Contrôleur à cames
	ZEL 22 0006	Bx9	115 V	Nockenschalter	Cam controller	Contrôleur à cames
	ZEL 22 0002	Bx9	220 V	Nockenschalter	Cam controller	Contrôleur à cames
4	ZEL 40 1010	GHV 87 10 603		Not-Aus-Taste	Emergency shut down	Arrêt d'urgence
5	ZEF 01 0060	LZ 60		Grobfilter	Filter	Filtre
6	ZMO 78 9115	4530 N	115 V	Lüfter	Fan	Ventilateur
	ZMO 78 9220	4580 N	220 V	Lüfter	Fan	Ventilateur
7	ZKO 24 0600	60MF 400V-DB	100 V	Motorkondensator	Motor-capacitor	Condensateur de moteur
	ZKO 24 0500	50MF 400V-DB	115 V	Motorkondensator	Motor-capacitor	Condensateur de moteur
	ZKO 24 0120	12MF 400V-DB	220 V	Motorkondensator	Motor-capacitor	Condensateur de moteur

COMPACT 5 PC
Verdrahtungsplan wiring diagram

Pos.	Ref. No.	DIN		Benennung	Description	Designation
	V3U 176			Dreibackenfutter	3-jaw-chuck	Mandrin à 3 mors
1	ZSR 12 0425	M4x25 DIN 912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
2	V1A 000 040			Deckel	Cap	Couvercle
3	ZSR 12 0540	M5x40 DIN 912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
4	V1A 000 030			Triebling	Pinion	Pignon
5	V1A 000 020			Zahnkranz	Scroll	Couronne
6	V1A 000 010			Frontplatte	Front plate	Feuille frontal
7	V1A 000 060			Sicherungsschraube	Retaining screw	Vis de retenue
8	ZME 31 0843			Satz Außenbacken	Outside stepped jaws	Jeu de mors extérieurs
	ZME 31 0853			Satz Innenbacken	Inside stepped jaws	Jeu de mors intérieur
9	V1A 000 070			Schlüsselteil	Key	Cle
10	A5Z 410 060			Knebel	Toggle	Garrot
	V3U 178			Vierbackenfutter	4-jaw-chuck	Mandrin à 4 mors
1	ZSR 12 0425	M4x25 DIN 912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
2	V1B 000 040			Deckel	Cap	Couvercle
3	ZSR 12 0540	M5x40 DIN 912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
4	V1A 000 030			Triebling	Pinion	Pignon
5	V1A 000 020			Zahnkranz	Scroll	Couronne
6	V1B 000 010			Frontplatte	Front plate	Feuille frontal
7	V1A 000 060			Sicherungsschraube	Retaining screw	Vis de retenue
8	ZME 31 0844			Satz Außenbacken	Outside stepped jaws	Jeu de mors extérieurs
	ZME 31 0854			Satz Innenbacken	Inside stepped jaws	Jeu de mors intérieur
9	V1A 000 070			Schlüsselteil	Key	Cle
10	A5Z 410 060			Knebel	Toggle	Garrot

Pos.	Ref. No.	DIN		Benennung	Description	Designation
	2oo o4o			Spannzangenvorrichtung	Collet holder	Porte-pinces
1	A5Z o4o o1o			Zangenhalter	Collet holder	Porte-pinces
2	A5Z o4o o2o			Spannmutter	Locking ring	Ecrou de serrage
3	ZST 11 o71o	6x1oo DIN 7		Zylinderstift	Dowel pin	Tige de serrage
4	ZSR 33 o51o	M5x1o DIN 933-5.6		Sechskantschraube	Hexagon head screw	Vis hexagonale

Pos.	Ref. No.	DIN		Benennung	Description	Designation
	2oo 36o			Aufspannscheibe	Clamping plate	Plateau de serrage
1	A3Z 35o o2o			Spannschraube 55	Clamping screw 55	Vis de serrage 55
2	A3Z 35o o3o			Spannschraube 35	Clamping screw 35	Vis de serrage 35
3	A2Z 351 o1o			Spanneisen	Clamping jaw	Bride de serrage
4	ZSR 33 o64o	M6x4o DIN 933-5.6		Sechskantschraube	Hexagon head screw	Vis hexagonale
5	ZSR 33 o62o	M6x2o DIN 933-5.6		Sechskantschraube	Hexagon head screw	Vis hexagonale
6	ZSR 12 o52o	M5x2o DIN 912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
7	ZSB 25 o64o	B6,4 DIN 125		Scheibe	Washer	Rondelle
8	ZMU 34 o6oo	M6 DIN 934-6		Sechskantmutter	Hexagon nut	Ecrou 6 pans

Pos.	Ref. No.	DIN		BENENNUNG	DESCRIPTION	DESIGNATION
	200 420			**Planscheibe**	**Independend chuck**	**Plateau americain**
1	A5Z 420 010			Gehäuse	Housing	Corps
2	A5Z 420 020			Satz von 4 Umkehrbacken	Set of 4 reversible jaws	Jeu de 4 mors reversible
3	A2Z 420 020			Spannschraube	Spindle	Broche
4	A2Z 420 040			Schlüssel	Key	Cle
5	ZST 75 0460	4x60DIN1475-6.8		Knebel	Toggle	Garrot
6	ZSR 12 0525	M5x25DIN912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux

Pos.	Ref. No.	DIN		Benennung	Description	Designation
	26o 31o			Gruppe Plotter	Plotter complete	Ens. Plotter
1	A6Z o11 ooo			Auflage	Table	Tableau
2	A6Z o1o o31			Halter	Bar	Support
3	A6Z o1o o5o			Führung	Holder	Guidage
4	A6Z o1o o6o			Leiste	Gib	Lardon
5	ZST 51 o4o4	M4x4 DIN 551-5.8		Gewindestift	Set screw	Vis pointeau
6	ZNA 76 0142	1,4x2,5 DIN1476-4.6		Kerbnagel	Grooved drive stud	Clou cannelé
7	ZRG 71 141o	W14x1 DIN 471		Sicherungsring	Retaining ring	Anneau de retenue
8	ZSR 64 o515	M5x15		Rändelschraube	Knurled screw	Vis moletée
9	A6Z o1o o4o			Achse	Axis	Axe
10	ZRO o6 7o7o	7o x 7o		Papierrolle	Paper roll	Rouleau à papier
11	ZSR 12 o525			Zylinderschraube	Socket head screw	Vis 6 pans creux
12	ZSR 12 o812	M8x12 DIN 912-6.9		Zylinderschraube	Socket head screw	Vis 6 pans creux
13	ZST 99 1ooo			Plotterstift	Plotter pen	Crayon Plotter

B SOFTWARE DESCRIPTION

1. VOCABULARY EXPLANATIONS

bit (b)	Binary sign = smallest memory unit of the computer (e.g., current - not current or magnetic - not magnetic)
Byte (B)	Memory unit to store a letter, a number or a sign (1 byte = 8 bit) Kilobyte (K, kB): usual unit (measurement) for the memory capacity of a computer (1 kilobyte = 2^{10} bytes = 1024 bytes) Megabyte (MB): 1 megabyte = 1024 kilobytes
CAD	Computer Aided Design = tool construction (see chapter 4.2)
CAM	Computer Aided Manufacturing = determining the machining cycle (see chapter 4.3)
CNC	Computer Numerical Control
Coprocessor	Additional arithmetic unit as a support for the microprocessor
Disk	Exchangable, magnetic plastic disk, to store software (memory capacity, e.g., 5¼ inches disk with 360 + kB or 1.2 MB or 3½ inches disk with 720 kB or 1.44 MB)
Hard disk	Magnetized disks permanently located in the computer, to store software (memory capacity for PC, e.g., up to 80 MB)
Graphics card	Shapes the computer signals for the screen
Hardware	Are all usable objects
Compatibility	Two devices are compatible, if they can operate with each other without any additional equipment or can be exchanged for each other
Menu	List of the possibilities offered
Microprocessor	Central arithmetic unit of the PC
NC program	List of all information, which a machine tool needs to manufacture a work piece
PC	Personal Computer = multi-purpose, small computer for one person
Postprocessor	Converts a program into information, which the CNC machine tool understands and receives
RAM	Random Access Memory = operational memory = component, which loses information, when the device is switched off (memory capacity of the RAM usually amounts to 640 + kB for PC)
Interface	Connection element for connecting external devices to the PC Serial interface: processes 1 bit Parallel interface: processes 1 byte (= 10-fold information flow speed)
Software	The software is a thought-defined product (e.g., a program on a disk), which controls the computer. System software: basic, administrative program; contains the operating system (e.g., DOS = Disk Operating System) and facilitates the running of the application software Application software: special operational program (e.g., Unimat PC software). The hardware can carry out the tasks given by this program.

2. SOFTWARE INSTALLATION , ACCESS INTO PROGRAM

2.1 Possible PC configuration

In order to operate with this EMCO software, you require a PC with the following capabilities:

- IBM compatible (PC, PC/XT, PC/AT, PC/AT-386)
- Operating system MS-DOS from version 3.1
- 640 kB RAM (main memory)
- 1 Hercules graphics card with a parallel interface (Centronics) or 1 + EGA colour graphics card
- Monochrome screen or EGA colour screen capable of graphics
- 2 disk drives or 1 disk drive + 1 hard disk with 20 MB
- 1 parallel interface for the use of an EGA colour graphics card (for the connection of a printer = option)

An arithmetic coprocessor is not absolutely necessary for the operation with this EMCO software.
This coprocessor is, above all, recommended for PCs with a slow processing speed (e.g.: PC/XT with a cycle frequency of 4.77 or 8 MHz), however, since it executes the arithmetic operations much faster.
The use of a coprocessor is not required for PCs with a cycle frequency of over 10 MHz (PC/AT).

Example: IBM compatible PC/AT with
- 640 kB RAM
- 1 disk drive 1.2 MB
- 1 hard disk 44 MB
- 1 EGA graphics card
- EGA screen capable of graphics
- keyboard
- operating system MS-DOS version 3.3
- 1 parallel interface

Compatibility problems
The software runs on all 100 % IBM compatible PCs. Upon request, EMCO informs you of all those processers, on which this EMCO software was tested.

2.2 PC with hard disk and 1 disk drive

Installation

Start DOS (Disk Operating System). If necessary, consult the DOS manual, which was supplied with your computer.

The files INT10.COM and HGC.COM or HGC.EXE must be available on your hard disk (in the main index) and be called up when using a Hercules graphics card in order to initialize this card.
These files may not be supplied by us due to copyright laws.
If you install this EMCO software on a PC with a Hercules graphics card, the recall of these files is automatically written into the file AUTOEXEC.BAT during the installation of the software.

When the DOS prompt "C: \ >" appears, insert the installation disk 1 into the drive A.

Enter the following command:

Screen display	Entry	Explanation
C:\ >	A : ↵	Change to drive A
A:\ >	I ↵	Call up installation program

After entering the command "I" and pressing the ENTER key, the registration screen for setting the language appears.

Setting the language

Move the light beam with the arrow keys to the desired language and confirm it with .

Entering the program path (sub-directory)

This EMCO software is installed in the sub-directory COMPACT by pressing .

Choose printer

Choose the right printer of your work station.

If you don't use a printer, just press .

Altering the program path:

You can also install this EMCO software in a sub-directory determined by you.

Procedure:

- You can delete the suggested sub-directory name letter by letter with the key .
- Enter a new sub-directory name.

Example:

C: \ training \ .

This EMCO software is installed in the sub-directory "training"

Attention:

Do not forget \ after entering the sub-directory name.

Control menu:

```
emco
COMPACT 5-PC INSTALLATION 1.0

CONFIGURE        IBM GRAPHIC
  UNIT             METRIC
 PATH            C:\COMPACT\

Are specifications correct ?

      YES              NO
```

The previous specifications are displayed:

YES ... The previous specifications are correct, the software is installed.
NO .... The previous specifications are incorrect, the installation begins again with "Entering the type of PC".

Select the desired menu point with the arrow keys and confirm it.

If the menu point YES was selected and confirmed, the following appears on the screen:

```
emco
COMPACT 5-PC INSTALLATION 1.0

The COMPACT 5 PC SOFTWARE is
 compiled and copied into
   the given path ...

     Please wait !

Copying ...

C:\COMPACT\
```

The individual files are copied from the drive A into the sub-directory determined by you.

The following message appears after a short time:

```
Disk 2 in drive A:    insert
```

Remove the installation disk 1 from the drive A and insert the installation disk 2 and confirm it.

The following appears:

```
Copying ...  5 - 10 minutes
```

If the software was installed correctly in the PC, the following appears:

```
emco

Installation is finished
```

The PC registers with drive A.

Changing the drive:

Screen display	Entry
A:\>	C : ↵
C:\>	

The software can be called up (see next page).

Access

After the DOS prompt "C: \ >" appears, enter the following commands:

Screen display	Entry
C : \ >	C O * COMPACT ↵
C :\COMPACT >	

* Space bar

In this way, you change to the sub-directory COMPACT.

Calling up the program:

If you have altered the program path (= sub-directory) during the installation of the software, you must enter your selected sub-directory name (here COMPACT).

Screen display	Entry
C :\COMPACT >	C O M P A C T ↵

The title screen appears:

Pressing ENTER:

You are now in the program.

3. EXPLANATIONS FOR THE SOFTWARE

3.1 The differences in the input keyboards

Small PC keyboard

Large PC keyboard

The same key functions are arranged and designated differently on these two types of keyboard.

Differences in the key designation

Small PC keyboard	Large PC keyboard
9 PgUp ↑	PgUp ↑
3 PgDn ↓	PgDn ↓
7 Home	Pos 1
1 End	End
0 Ins	Ins
. Del	Del
Prt sc *	Prt sc

Note
The keys of the large PC keyboard are used in these instructions.

3.2 Important key functions

SHIFT key
If the SHIFT key is pressed at the same time as the desired key, the upper case of the respective key is selected.

Example
If the SHIFT key is not active, the lower case (+) is selected.

Shift If the SHIFT key is pressed, the upper case (*) is selected.

Note
Only the selected key case is shown in these instructions for easier reading.

Preceding zeros
Following zeros
Zeros before a number and after the comma can but do not need to be entered.

Plus/minus sign
Plus signs (+) are not entered. The minus sign (-) must be entered before the number.

Entering large, small letters
Letters can be entered as large or small letters.

Enter key
Inputs are confirmed or stored with the "ENTER" key.

Decimal point
The decimal point is entered with this key.

Separating two entries
Two subsequent entries are separated with the comma key.

Example
For a work piece compilation, the PC requests:

Diameter, work piece length?

Input: 23.5 , 46

2 3 . 5 , 4 6 ↵

Error correction
The entry can be corrected with this key. The previously entered character is deleted.

Interrupting selected commands
Esc The selected command is interrupted with this key.
The previously selected menu level can also be returned to with the Esc. key.

3.3 Coordinate system

The piece of information "Move the longitudinal slide towards the head stock" is a long piece of information; it would also be different in every language. Thus, the directions of movement for NC machines are described in a coordinate system.

Coordinate system of the machine

Z axis = Axis parallel to the rotary axis

X axis = Axis perpendicular to the rotary axis

- Z movement = Movement of the longitudinal slide towards the head stock

+ Z movement = Movement of the longitudinal slide away from the head stock

+ X movement = Movement of the cross slide away from the rotary axis

- X movement = Movement of the cross slide towards the rotary axis

Representing the coordinate system on the screen

3.4 Screen cursor

A cursor (graticule) is shown on the screen. This cursor can be moved on the screen.

Moving the screen cursor
The cursor is moved on the screen by means of the 4 arrow keys.

Further key functions

Doubling the step size of the cursor

Halving the step size of the cursor

The cursor jumps to the middle of the screen

The cursor jumps to the reference point

Step size .... distance of movement each time the key is pressed

Showing the screen cursor
There are two ways of showing the cursor on the screen:

1. Cursor as a graticule
   Normal representation of the cursor

2. Reduced cursor
   A reduced cursor is shown, when the key "<" is pressed.

The cursor is shown as a graticule again, when the key ">" is pressed.

Cursor programming methods
You can describe the path of the screen cursor in the program: there are generally two ways of describing this path.

Absolute value programming (reference value programming)
The points, to which the screen cursor is to proceed, are given starting from a zero point (reference point R).

Example

Incremental value programming
Chain dimensions are entered for the incremental value programming. The currect position of the screen cursor is the starting point for every path description.

Example

3.5 Menu structure

This EMCO software is divided into menus (= selection possibilities). Divisions into main menu, menus and submenus are used depending on the hierarchy. Jumps back- and forwards between these individual menus can be made without limitations.

F? → Go to the next menu level

Esc. → Return to the previous menu level

Menu summary

Notes

You can select and activate the individual menu points with the function keys F1 to F10 or with the space bar.

A command can be interrupted or a return to the previous menu level activated with the key Esc.

3.6 Screen display

1 Diagrammatic representation of the working area
2 Displaying the cursor position [mm]
  X, Z ... Position of the cursor
  ::: ... Step size of the cursor (distance travelled every time a key is pressed)
3 Displaying the tool position
  X, Z ... Position of the tool [mm]
  D ... Diameter position of the tool [mm]
  F ... Feed speed [mm/min]
  S ... Main spindle speed [rpm]
4 Displaying the screen messages
5 Menu display

3.7 Selection / deselection of the menu points

There are two possibilities:

Possibility 1
By means of the space bar and ↵

Example
Activating the PROGRAM menu

	Go to the desired menu point with the space bar
↵	Confirm entry and activate PROGRAM menu
Esc	Return to the main menu

Possibility 2
By means of the function keys F1 to F10

Example
Activating the PROGRAM menu

F3	Selection and activation of the PROGRAM menu
Esc	Return to the main menu

PROGRAM

F1	RAPID
F2	FEED
F3	CYCLES
F4	EDITOR
F5	TOOL CHNGE
F6	ERASE LAST
F7	ERASE PROG
F8	FEED
F9	G94/G95
F10	OFFSET

Note
The activation of the individual menu points is described with the function keys in these instructions.

4. MENU POINTS

4.1	Work piece	54
4.2	Draw	55
4.2.1	Cursor	56
4.2.2	Point	59
4.2.3	Line	61
4.2.4	Arc	64
4.2.5	Visible edges	68
4.2.6	Mirroring	68
4.2.7	Cleaning	68
4.2.8	Erasing the geometric element	69
4.2.9	Erasing all	69
4.2.10	Hotkeys - draw	70
	Specimen example 1	71
	Specimen example 2	72
	Specimen example 3	73
4.3	Program	74
4.3.1	Moving in the rapid	74
4.3.2	Moving with feed speed	74
4.3.3	Cycles	74
	Roughing cycle	75
	Back pocket cycle	77
	Follow contour cycle	79
	Cut-in cycle	81
	Threading cycle	83
4.3.4	Edit (see chapter 6)	90
4.3.5	Tool change	91
4.3.6	Erase the last movement	91
4.3.7	Erasing the machining program	91
4.3.8	Entering the feed values	91
4.3.9	Change feed unit	91
4.3.10	Entering the finishing offsets	91
4.4	Display	92
4.4.1	Zoom all	92
4.4.2	Zoom window	92
4.4.3	Zoom work piece	93
4.4.4	Removing, inserting the head stock	93
4.4.5	Removing, inserting the work piece	93
4.4.6	Removing, inserting the tool	93
4.4.7	Removing, inserting the tool path	94
4.4.8	Inserting, removing the tail stock	94
4.4.9	Changing the simulation display	94
4.4.10	Hotkeys - machine	95
4.5	Set up 0,0	96
4.6	Manual	97
4.7	Machine	98
4.7.1	Automatic	98
4.7.2	Single	98
4.7.3	Fast run	98
4.7.4	Empty cut step	98
4.7.5	Repeating the program	99
4.7.6	Moving the tool to the cursor	99
4.7.7	Shifting the program	99
4.7.8	Altering the scale program	99
4.8	Archive	
4.8.1	Storing the program	100
4.8.2	Loading the program	100
4.8.3	Storing the geometry	101
4.8.4	Loading the geometry	101
4.9	Print	101
4.10	End	101

4.1 Work piece (F1)

The work piece is defined in this menu, and it is displayed clamped in the chuck.

Unit: [mm]

Example

F1

Screen message

Diameter, work piece length:

22,100 ↵

Diameter, length of the drill hole (ENTER = no)

no ↵

yes 4,10 ↵

The work piece clamped in the chuck appears on the screen, and the exit from the WORK PIECE menu takes place at the same time.

Note
The reference point = R (point X = 0, Z = 0 of the coordinate system) is situated on the Z axis at the right-hand end of the work piece, when the work piece is called up.

4.2 Draw (F2)

The contour of the work piece is determined in this menu.

Preliminary explanations

You only have to draw the lower half of the work piece contour on the screen, without circumferential edges. Circumferential edges are added after drawing the work piece contour, and by mirroring around the Z + axis, you automatically receive the complete representation of the work piece contour.
The compilation of the work piece contour is carried out in the submenus described on the following pages.

Summary of sub menus

Description of the individual sub-menus

4.2.1 Cursor (F1)

In this submenu, it is determined, how the cursor is to be positioned on the screen.

CURSOR			
F1	CURSOR INC	F6	STEP SIZE
F2	CURSOR ABS	F7	TO REF PNT
F3	CURSOR R, A	F8	MID SCREEN
F4	CURSOR X, A	F9	INTERSECT
F5	CURSOR Z, A	F10	

Moving the screen cursor
The cursor is moved on the screen by means of the 4 arrow keys.

Further key functions

PgUp ↑ Doubling the step size of the cursor

PgDn ↓ Halving the step size of the cursor

Pos 1 The cursor jumps to the middle of the screen

End The cursor jumps to the reference point

Step size .... distance of movement each time the key is pressed

Showing the screen cursor
There are two ways of showing the cursor on the screen:

1. Cursor as a graticule
   Normal representation of the cursor

2. Reduced cursor
   A reduced cursor is shown, when the key "<" is pressed.

The cursor is shown as a graticule again, when the key ">" is pressed.

Calculating functions in the CURSOR submenu
For every menu point in the CURSOR submenu, where a number is prompted, there are many calculating functions available to you for the entry of arithmetical functions. In this way, you possess an aid capable of performance, which gives valuable help, especially for coordinate entries and angle calculations.

The following signs or mathematical functions can be used:

Entry	Function
+	Addition
-	Subtraction
*	Multiplication
:	Division
**	To the power of
(	Open bracket
)	Close bracket
SIN	Sin
COS	Cos
TAN	Tan
DSIN	Sin in degrees
DCOS	Cos in degrees
DTAN	Tangent in degrees
ASIN	Arc sin
ACOS	Arc cos
ATAN	Arc tangent
DASIN	Arc sin in degrees
DACOS	Arc cos in degrees
DATAN	Arc tangent in degrees
SQRT	Square root
PI	3.1415927

Any arithmetical function can be entered instead of a simple number for numerical entries (= numbers). The use of brackets is allowed. These can be nested as much as is required.

Note
Entered functions must be written in brackets (see example).

Example
You are in the CURSOR submenu. An incremental coordinate is to be calculated.

$X = 10\frac{\sqrt{3}}{2}$, $Z = 50$

Therefore, enter the following:

F1

10 * SQRT (3) : 2, 50 ↵

A cursor step of 8.66 in X and 50 in Z is the result.

Positioning the cursor incrementally (F1)

Unit: [mm]
The cursor is shifted from the present position by the entered values.

Positioning the cursor absolutely (F2)

Unit: [mm]
The cursor is shifted by the values entered (always measured from the reference point).

Positioning the cursor with the radius and angle (F3)

Unit: [mm], [°]
The cursor is shifted from the present position by the radius and angle entered.
Example: R = 24 mm, A = -30° (clockwise)

Screen message	Entry
	F3
Radius, angle movement from the cursor?	24, -30 ↵

Note: Angle entry
Positive (+) sign = angle in the anti-clockwise direction.
Negative (-) sign = angle in the clockwise direction.

Positioning the cursor with X dimension and angle (F4)

Unit: [mm], [°]
Entering the X dimension incrementally

Example: X = 19 mm, A = - 45°

Screen message	Entry
	F4
X, angle movement from the cursor?	19, - 45 ↵

Positioning the cursor with Z dimension and angle (F5)

Unit: [mm], [°]
Entering the Z dimension incrementally

Example: Z = 15 mm, A = -35° (clockwise)

Screen message	Entry
	F5
Z, angle movement from the cursor?	15, - 35 ↵

Determining the step size of the cursor (F6)

Unit: [mm]
The step size is the distance the cursor moves every time a key is pressed.

The step size of the cursor is 1 mm when you call up the software.

Example: Cursor step should amount to 2.5 mm

Screen message	Entry
	F6
Cursor step 1.000 New step (ENTER = no change)	2,5 ↵

Cursor to the reference point (F7)

The reference point is the origin of the coordinate system shown on the screen (reference point = position X = 0 / Z = 0 of the coordinate system).

Position of the reference point
No work piece:
The reference point is situated on the Z axis at the stop of the clamping device.

Workpiece compiled:
The reference point is situated on the Z axis at the right-hand edge of the work piece.

The cursor jumps to the reference point, when the key F7 is pressed.

Cursor to the middle of the screen (F8)

The cursor jumps to the middle of the screen, when the key F8 is pressed.

Find the point of intersection (F9)

Position the cursor by means of the arrow keys near to the point of intersection to be found.

The cursor jumps to the neighbouring point of intersection, when the key F9 is pressed.

Possibilities for the point of intersection
- two lines
- two circles
- a line and a circle

4.2.2 Point (F2)

Secondary points for drawing the work piece contour are determined on the screen in this submenu.
The basis of the definition of the geometry (= contour of the work piece) are secondary points.
E.g., a line is defined by a starting point and an end point.

	POINT
F1	START PNT
F2	END POINT
F3	CENTRE PNT
F4	FIND POINT
F5	NEW REF.PT
F6	TO REF PNT
F7	INTERSECT
F8	TRIM BITS
F9	
F10	

Determining the starting point (F1)

The point, on which the cursor is located, is defined as the starting point of an element of geometry (line, circle or arc) by pressing the key F1.

The active starting point is shown on the screen by the symbol " > ".

Determining the end point (F2)

The point, on which the cursor is located, is defined as the end point of an element of geometry (a line, circle or arc) by pressing the key F2.

The active end point is shown on the screen by the symbol " < ".

Determining the centre point (F3)

The point, on which the cursor is located, is defined as the centre point of a circle or an arc by pressing the key F3.

The active centre point is shown on the screen by the symbol "o".

Representation of a starting point, centre point and end point

Find point (F4)

Position the cursor by means of the arrow keys near to the point found.
The cursor jumps to the neighbouring point, when the key F4 is pressed.

Purpose:
If machining is to continue from an available point, this must be approached exactly with F4.

Example:
The cursor is located at the starting point of the line.

Re-defining the reference point (F5)

The point, on which the cursor is located, is defined as the new reference point by pressing the key F5.
Reference point (R) = origin of the coordinate system (X = 0 / Z = 0).

Cursor to the reference point (F6)

The cursor returns to the reference point, when the key F6 is pressed.

Find the point of intersection (F7)

Position the cursor by means of the arrow keys near to the point of intersection to be found.

The cursor jumps to the neighbouring point of intersection, when the key F7 is pressed.

Trim bits (F8)

Trim bits = deleting protruding parts of an element of geometry

Example

Two intersecting lines should result in a bit.

Position the cursor by means of the arrow keys near to the bit to be trimmed.

4.2.3 Line (F3)

Lines are drawn, altered or erased in this submenu.

LINE	
F1	DRAW
F2	SEARCH
F3	SPLIT
F4	ERASE
F5	PARALLEL
F6	PERPENDIC
F7	TANGNT P/A
F8	TANGNT A/A
F9	CHAMFER
F10	

Drawing a line (F1)

A line is drawn, whose starting and end points are already determined.

Example
Starting point: X = 35 mm, Z = 65 mm
End point: X = 25 mm, Z = 55 mm

A line between the two points is drawn by pressing the key F1.

Searching for a line (F2)

Position the cursor by means of the arrow keys near to the line to be searched for.
The cursor jumps to the neighbouring line, when the key F2 is pressed.

Purpose: If a line is to be machined further, this must first be determined with F2.

Splitting a line (F3)

Position the cursor by means of the arrow keys on the splitting point of the line.

This line is divided into two lines, when the key F3 is pressed. (The point of splitting is invisible.)

Purpose: If only a part of a line is to be machined further, this part must be split from the rest.

Erasing a line (F4)

Position the cursor by means of the arrow keys near to the line to be erased and press the key F4.

Drawing a parallel line (F5)

Unit: [mm]
A parallel line is drawn to an existing line (e.g., distance = 5 mm).

Drawing a perpendicular (90°) to a line (F6)

A perpendicular (90°) is drawn to an existing line.

Move the cursor to the point, where the perpendicular should start.

F6

Screen message

Position cursor on the desired line

Move the cursor near to the line.

Drawing a tangent point/arc (F7)

A tangent from the current cursor position to an arc is drawn.

F7

Screen message

Position cursor on the desired arc.

Move the cursor to the desired arc.

Is this the desired tangent (ENTER = yes)

yes

no

Any key except for Esc. or ↵

Is this the desired tangent (ENTER = yes)

yes

no

Any key except for Esc. or ↵

Drawing a tangent arc/arc (F8)

A tangent from an arc to a second arc is drawn.

F8

Screen message

Position cursor on the desired arc

Move the cursor to the 1st arc.

Position cursor on the desired arc

Move the cursor to the 2nd arc.

Is this the desired tangent (ENTER = yes)

yes

no

Any key except for Esc. or ↵

Is this the desired tangent (ENTER = yes)

yes

no

Any key except for Esc. or ↵

Adding a chamfer (45°) (F9)

Unit: [mm]

Condition:

1. Both lines contact each other at this point (do not intersect).
2. The cursor must be positioned near to this bit.

Example: chamfer = 3 x 3 mm

Screen message	Entry
	F9
Size of the chamfer	3 ↵
Is the chamfer correct (ENTER = yes)	↵

Possibilities:

ENTER = Confirming the desired chamfer

Any other key = interruption of this command "Adding a chamfer".

4.2.4 Arc (F4)

Arcs are drawn, altered or erased in this submenu.

	ARC
F1	DRAW
F2	SEARCH
F3	SPLIT
F4	ERASE
F5	CIRCLE
F6	ARC 3 PTS
F7	FILLET
F8	FILL 2 ELE
F9	
F10	

Drawing an arc (F1)

An arc is drawn, whose starting, centre and end points are already determined.

Example:

Starting point:	X = 35 mm,	Z = 66 mm
Centre point:	X = 30 mm,	Z = 58 mm
End point:	X = 25 mm,	Z = 55 mm

An arc (clockwise) is drawn from the starting point to the end point by pressing F1.

Note:
Arcs are always drawn clockwise.

You can define the position of the arc by the selection of the starting and end points.

Searching for an arc (F2)

Position the cursor by means of the arrow keys near to the arc to be searched for.
The cursor jumps to the neighbouring arc, when the key F2 is pressed.

Purpose
If an arc is machined further, this must be defined with F2 first.

Splitting an arc (F3)

Position the cursor by means of the arrow keys on the splitting point of the arc.

The arc is split, when the key F3 is pressed. (The splitting point is not visible.)

Purpose:
If only a part of an arc is to be machined further, this part must first be split from the rest.

Application example:
You must split arcs over 90°, since a starting point of the back pocket must be determined in the program menu (for back pocket).

Erasing an arc (F4)

Position the cursor by means of the arrow keys near to the arc to be erased and press the key F4.

Attention!
The command "Erase" can no longer be reversed, after it has been carried out!

Drawing a complete circle (360°) (F5)

Unit: [mm]
Position the cursor by means of the arrow keys on the centre point of the desired complete circle.
Press F5, enter the radius of the circle and confirm it.

Example:
Constructing a complete circle R = 15 mm

Screen message	Entry
	F5
Circle radius 10.000 (ENTER = no change):	15 ↵

Drawing an arc with three points (F6)

Determine the starting point and the end point.
Position the cursor on the circumference of the desired arc (= third point).
An arc is drawn through the starting, end point and cursor position, when the key F6 is pressed.

Note:
Arcs are always drawn anti-clockwise.
You can define the position of the arc by the selection of the starting and end points.

Example:
Starting point: X = 25 mm, Z = 36 mm
End point: X = 16 mm, Z = 25 mm
Cursor position: X = 18 mm, Z = 32 mm

The arc is drawn, when F6 is pressed.

Adding a fillet (F7)

Unit: [mm]
1. Both lines must contact at this point.
2. The cursor must be positioned near to this bit.

Example: Radius = 5 mm

Screen message	Entry
	F7
Radius for the corner?	5 ↵
Is the radius correct (ENTER = yes)	↵

Possibilities:
ENTER = confirming the desired radius
Any other key = interruption of this command "Adding the fillet".

Fill two elements (F8)

Geometric elements (points, lines, circles or arcs) are filled tangentially by an arc.

Note:
If you want to fill a circle or an arc, the prompt from the system for the type of filling appears. There are two possibilities.

Filling 1: in front of the circle

Filling 2: including the circle

Example 1
Filling the point of one line with a point of another line.

F8

Screen message

Filleting tangential to: 1 = point, 2 = line, 3 = arc:

1

Define element nr. 1

Position the cursor on the desired point.

Filleting tangential to: 1 = point, 2 = line, 3 = arc:

1

Define element nr. 2

Position the cursor on the desired point.

Radius for the corner? :

8

Position the cursor in the approximate centre

Move the cursor near to the centre point.

Is this the desired side? (ENTER = yes)

yes

no

Any key except for Esc. or

Example 2
Filling the point of a line with a line.

F8

Screen message

Filleting tangential to: 1 = point, 2 = line, 3 = arc:

1

Define element nr. 1

Position the cursor on the desired point.

Filleting tangential to: 1 = point, 2 = line, 3 = arc:

2

Define element nr. 2

Position the cursor on the desired line.

Radius for the corner? :

8

Position the cursor in the approximate centre.

Move the cursor near to the centre point.

Is this the desired side? (ENTER = yes)

yes

no

Any key except for Esc. or

Example 3
Filling the point of a line with an arc.

F8

Screen message

Filleting tangential to: 1 = point, 2 = line, 3 = arc:

1

Define element nr. 1

Position the cursor on the desired point.

Filleting tangential to: 1 = point, 2 = line, 3 = arc:

3

Filleting type? : 1 = before the arc, 2 = include the arc:

1

Define element nr. 2

Position the cursor on the desired circle.

Radius for the corner? :

6

Position the cursor in the approximate centre

Move the cursor near to the centre point.

Is this the desired side? (ENTER = yes)

yes

no

Any key except for Esc. or

Example 4
Filling a straight line with an arc.

F8

Screen message

Filleting tangential to: 1 = point, 2 = line, 3 = arc:

2

Define element nr. 1

Position the cursor on the desired line.

Filleting tangential to: 1 = point, 2 = line, 3 = arc:

3

Filleting type? : 1 = before the arc, 2 = include the arc:

1

Define element nr. 2

Position the cursor on the desired circle.

Radius for the corner? :

6

Position the cursor in the approximate centre

Move the cursor near to the centre point.

Is this the desired side? (ENTER = yes)

yes

no

Any key except for Esc. or

4.2.5 Drawing visible edges (circumferential edges) (F5)

Drawing in the visible edges is not required for the production on a lathe; it only serves to create a complete representation according to the standards.

Procedure:
- The cursor was positioned near to the desired visible edge.
- The visible edge is drawn up to the Z axis, when the key F5 is pressed.
- This procedure is repeated for every visible edge.

4.2.6 Mirroring the work piece contour (F6)

Only the lower half of the work piece must be drawn on the screen for the production on a lathe.
The upper half of the work piece can be created by mirroring around the Z axis.

Procedure
Press F6 key
The command F6 - mirroring remains. This means, that all contour parts programmed after this function are also mirrored.

The command is deselected again, when "F6" is pressed again.

4.2.7 Cleaning the work piece contour (F7)

At the beginning of a geometric definition, it will often be the case that two or more contour elements are cut uncleanly. For the machining of the work piece, however, continuous contours are required. The points of intersection must be "cleaned" (for example, see next page).

Note:
The prompt from the system for the type of contour appears during the cleaning. There are two possibilities.

Possibility 1: open contour

Possibility 2: closed contour

Example: Cleaning

4.2.8 Erasing the geometric element (F8)

The cursor was positioned near to the element be erased (line, circle or arc).
The neighbouring geometric element is erased, when the key F8 is pressed.

Attention!
The command "Erase" can no longer be reversed, after it has been carried out.

4.2.9 Erasing all (F9)

All the drawing elements shown on the screen are erased.

Attention!
The command "Erase" can no longer be reversed, after it has been carried out.

Screen message	Entry
	F9
Are you sure? (ENTER = yes)	↵

Possibilities:
ENTER = drawing is erased
Any other key = erasion command is interrupted

Specimen example 1

Creating a work piece with F1

Ø = 22 mm, length = 100 mm, no bore hole

The work piece is enlarged over the whole of the screen surface by means of the hotkey Z .

Creating the lower half of the work piece contour with F2

Adding the radii and chamfers with F2

Adding the visible edges with F2, submenu F5

Reflecting the work piece contour with F2, submenu F6

Storing the work piece contour with F8

Specimen example 2

Creating a work piece with F1
⌀ = 22 mm, length = 100 mm, no bore hole

The work piece is enlarged over the whole of the screen surface by means of the hotkey Z .

Adding the visible edges with F2, submenu F5

Creating the lower half of the work piece contour with F2

Reflecting the work piece contour with F2, submenu F6

Adding the radii and chamfers with F2

Storing the work piece contour with F8

Specimen example 3

Creating a work piece with F1
⌀ = 22 mm, length = 100 mm, no bore hole

The work piece is enlarged over the whole of the screen surface by means of the hotkey Z

Adding the visible edges with F2, submenu F5

Reflecting the work piece contour with F2, submenu F6

Creating the lower half of the work piece contour with F2

Storing the work piece contour with F8

Adding the radii and chamfers with F2

4.3 Program (F3)

The machining cycle is determined for a work piece drawn and simulated on the screen in this menu. The tool path is shown graphically. The machining program is compiled automatically.

PROGRAM	
F1	RAPID
F2	FEED
F3	CYCLES
F4	EDITOR
F5	TOOL CHNGE
F6	ERASE LAST
F7	ERASE PROG
F8	FEED
F9	G94/G95
F10	OFFSET

Possibilities PROGRAM

4.3.1 Moving in the rapid (F1)

Moving in the rapid is a movement without chip removal and only serves to position the tool. The slides move with the highest possible speed (200 mm/min).

Representation on the screen: — · — · — · —

The tool moves to the cursor position in the rapid, when the key F1 is pressed.

4.3.2 Moving with feed speed (F2)

The slides move with the programmed feed speed (= machining movement, see F9).

Representation on the screen: — — — — — —

The tool moves to the cursor position with the feed speed set to the time (see F9), when the key F2 is pressed.

4.3.3 CYCLES (F3)

PROGRAM	
F1	ROUGHING
F2	BACK POCKET
F3	CONTOUR
F4	CUT-IN
F5	THREAD
F6	
F7	
F8	
F9	
F10	

A cycle is a pointed tool movement, which is enclosed and composed of many parts.
A circle is composed of the rapid and feed movements or thread-cut movements.

S = Starting point
E = End point
V_F = Feed speed
V_E = Rapid speed

Roughing cycle (F1)

The roughing cycle is a movement sequence, where the "coarse contour" of the work piece is machined with the greatest possible chip removal.

If your work piece is designed in such a way that the diameter of the work piece contour decreases to the left, this is not taken into consideration by the roughing cycle. This work piece area must be back pocketed in its own machining procedure.

Condition
The work piece contour created in the menu draw may not be interrupted, since otherwise an interruption of the roughing cycle takes place.

Procedure

1. Position the tool in the rapid (F1) or with the feed speed (F2) on a sensibly selected starting point (A) of the cycle and press F3.

2. Determining the starting point

The starting point (S) of the roughing cycle must always be the outermost point on the work piece contour in the Z direction.

Examples: Starting point for roughing cycles

If F3 - roughing cycle is pressed, the point nearest to the cursor is suggested by the program as the starting point.
Possibilities

[↵] Confirm the starting point

[Any key] Another point is suggested as the starting point.

3. Determining the contour points

After the starting point has been determined, the contour points (1, 2, ..) of the work piece contour drawn must be determined. These contour points are suggested to you.

Possibilities

[↵] .. Confirm the contour point
[1] .. Display the next contour point in the opposite direction
[2] .. Omit the suggested contour point
[3] .. End of the contour point determination, if all contour points have + been made known to the program.

You can also confirm the points of a back pocket, but these are not taken into account by the roughing cycle and must be machined with back pocket cycles.

4. Entering the technological data

After the contour points have been determined, the following prompts appear on the screen one after the other.

Screen display	Entry example
Finishing offsets [mm] X = 0 , Z = 0	0.1, 0.2 ↵
In-feed per cut 1 mm	0.8 ↵
Feed speed 150 mm/min	↵

Either confirm the suggested values with ENTER or enter new values.

The values confirmed here are suggested for the following machining variants (back pocket, follow contour).

The data for X values refer to the radius and not to the diameter of the work piece.

After entering the feed speed, the finishing cycle is processed on the screen. If it is not processed, there are the following causes of errors:

1. No enclosed contour (line draw).
2. Incorrect definition of the contour points.
3. The finishing offset is larger than the contour to be machined.
4. The cutting depth is larger than the contour to be machined.

Example - Roughing cycle

Call up the specimen example 2 (from page 72). Position the tool near to the work piece. Position the cursor near to the starting point.

F1

Screen message

Define start point, (ENTER = accept) any other key = continue

yes ↵ — no: Any key except for Esc. or ↵

Contour points are suggested by the computer and must be confirmed with ↵

1 = opposite direction, 2 = omit point, 3 = end (ENTER = accept)

↵

↵

3

Offset in X, Z 0.000 0.000 (ENTER = no change):

0.1, 0.2 ↵

Cutting depth = 1 (ENTER = no change):

0.8 ↵

Feed = 150 (ENTER = no change):

↵

Back pocket (F2)

Machining a work piece area, which was not taken into consideration by the roughing cycle (work piece diameter decreases to the left).

Condition
The work piece contour created in menu draw may not be interrupted, since otherwise an interruption of the back pocket takes place.

Procedure
1. Position the work piece on a sensibly selected starting point (A) in the rapid (F1) or in the feed speed (F2) and press F4.

Note
If a cycle was previously machined, the following message appears on the screen:

New or last contour (ENTER = last):

↵ Re-machine the last defined contour.

Any key Defining a new contour.

2. Determining the starting point
The starting point (S) of the back pocket must always be the outermost point of the machined contour in the Z direction.

If F2 - back pocket is pressed, the point next to the cursor is suggested by the program as the starting point.
Possibilities:

↵ Confirm the starting point

Any key Another point is suggested as the starting point.

Examples - Starting point for back pockets

Special case for turning out an arc with two different tools

Procedure

- Split the arc (point T)
- Draw the auxiliary line (H) from the splitting point (T) to the starting point (S1).
  Reason: The tool would otherwise cut above the arc and collide.
- Put an additional point (P) on the arc (or on the line).
  Reason: In this way, you can determine whether the tool is to travel the path S1 - T (= arc) or the path S1 - P - T (= line) during the determination of the contour points.
- Carry out the back pocket with the right lateral tool.
  Starting point S1
  Following contour points ....T, P, S1

- Change the tool and move to the new starting point (S2).
- Machine the rest from T to S1 with the roughing cycle (there is no more back pocket).
  Starting point S2
  Following contour points ... T, P, S1

Attention!
Arcs (larger than 90°), which are back pocketed, must be split before the machining (see page 64).

Reason
The system only recognizes the starting and end points of the arc. A new starting point must be defined for the back pocket. (The starting point of the back pocket is the point, which has come into being, when the arc was split.)

3. Determining the contour points

After the starting point was determined, the contour points (1, 2,..) of the back pocket drawn must be determined. These contour points are suggested to you.

Possibilities

- [↵] ..... Confirm the contour point
- [1] ..... Display the next contour point in the opposite direction
- [2] ..... Omit the suggested contour point
- [3] ..... Finish the contour point determination

The last contour point of the back pocket must lie on the same level in the X direction or higher than the starting point.

4. Entering the technological data

This is carried out in the same way as described in the chapter "roughing cycle".

Example - Back pocket
The specimen example 2 (from page 72) is called up. The menu point roughing was carried out. Move the tool near to the starting point of the back pocket. Position the cursor near to the starting point.

[F2]
↓
Screen message

New or last contour (ENTER = last):

Any key except for Esc. or [↵]

Define start point, (ENTER = accept), any other key = continue:

yes: [↵] → Contour points are suggested by the computer and must be confirmed with [↵]

no: Any key except for Esc. or [↵]

1 = opposite direction, 2 = omit point, 3 = end, (ENTER = accept):

[↵]
⋮
[↵]
[3]

Offset in X, Z 0.100 0.200 (ENTER = no change):

0.2, 0.1 [↵]

Cutting depth = 0.8 (ENTER = no change):

[↵]

Feed = 150 (ENTER = no change):

[↵]

Follow contour cycle (F3)

If a finishing offset was determined in the menu points roughing or back pocket, this is machined with the follow contour cycle.

Finishing = Dimension-accurate finishing of the work piece contour

Procedure

1. Position the tool near to the starting point (S) in the rapid (F1) or with the feed speed (F2) and press F3.

Note
If a cycle was previously machined, the following message appears on the screen:

New or last contour (ENTER = last):

[↵] Re-machine the last defined contour.

[Any key] Define the new contour.

2. Determining the starting point

The starting point of the follow contour cycle must always be the outermost point on the machined contour in the Z direction.

S ..... Starting point

If F3 - follow contour is pressed, the point nearest to the cursor is suggested as the starting point.

Possibilities

[↵] Confirming the starting point

[Any key] Another point is suggested as the starting point.

3. Determining the contour points (1, 2, 3, 4, 5)

After the starting point has been determined, the contour points of the work piece contour drawn must be determined. These contour points are suggested to you.

Possibilities

[↵] ..... Confirming the contour point

[1] ..... Displaying the next contour point in the opposite direction

[2] ..... Omitting the suggested contour point

[3] ..... Ending the contour point determination

Note
Only the points of the machining, which can be finished with the selected tool, are confirmed as contour points.

E.g., the holes (point 3, 4, 7, 8) may not be confirmed.

4. Entering the technological data

The input of the finishing offset and feed speed is carried out exactly as described in the chapter "roughing cycle".

Cut-in cycle (F4)

With this cycle a cut-in can be produced.

Procedure:

1. Select cut-in tool
   (see menu PROGRAM/CHANGE TOOL - F5)

2. Place the cut-in tool with rapid motion (F1) or feed speed (F2) at the level (direction X) of the initial point of the cycle (A) and press F3.

   Note:
   You have to place the cut-in tool only in direction X at the initial point (A) of the cut-in cycle. Before working-off the cycle the tool moves automatically to the initial point in direction Z.

3. F4 - select cut-in cycle

4. Determine the starting point
   If the key F4 was pressed, the following message appears on the screen:

Identify starting point for cut-in.

By using the cursor keys move next to the starting point (S) and confirm with ENTER.

5. Determine final point:
   After determining the starting point the following message appears:

Determine final point for cut-in:

By using the cursor keys move next to the final point (E) and confirm with ENTER.

6. Enter technological data:
   After having determined the starting and the final point of the cut-in off the following inquiries appear on the screen in series:

Screen message	Input
Enter cutter width	3 ↵
Enter feed in mm/min = 150	70 ↵
Enter infeed depth	1 ↵

7. Confirm inputs:
   After entering the technological data the following message appears on the screen:

End of input (ENTER = accepted):

Possibilities:

↵ The cut-in cycle is worked off.

Any key Exit

Example cut-in cycle:

Training example 3 is called up. Menu point 3 roughing and contour sequences have been executed. The cut-in tool has been selected and moved next to the starting point of the cycle.

F4

Identify starting point for cut-in

Move cursor to the starting point and confirm with ↵

Enter final point for cut-in

Move cursor to the final point and confirm with ↵

Enter width of the cutter

3 ↵

Enter feed in mm/min = 150 (ENTER = unchanged):

70 ↵

Enter depth of advance

1 ↵

End of input (ENTER = accepted)

↵

Threading cycle (F5)

Possible threads:

- With this machine you can tap and thread right-hand and left-hand threads.
- Possible thread pitches:
  PC/AT: 0,5 - 3 mm (0,025" - 0,11")
  PC/XT: 0,5 - 1,5 mm (0,025" - 0,062")
- Metric threads:
  PC/AT: M3 - M24
  PC/XT: M3 - M10
- Inch threads:
  PC/AT: 0,112" - 3/4"
  PC/XT: 0,112" - 3/8"
- Flank angle 60° (with threading tool supplied)
- Speed: Smallest speed range

Theoretical knowledge

Function of thread-cutting

At each workpiece revolution the threading tool has to advance at a certain length (= threading pitch P). For this the main spindle and the slide feed have to be synchronized.

Finding of thread turn
A thread is always cut in several operations. At the start command for thread-cutting the main spindle must have a certain angular position. At the start command for the next screw the main spindle must have the same angular position, so that the threading tool finds its way into the thread turn.

Synchronization of speed and feed

Hand-operated lathe
With a hand-operated lathe the slide feed is carried out from the main spindle via toothed belts, translating gear wheels and leadscrew to the longitudinal slide. There is a closed, mechanical power flux. If the main spindle turns slower, e.g. because of load, the main spindle turns slower at the same rate. The thread pitch always remains the same.

CNC lathe
At the spindle there is an aperture disk with light barrier (= encoder). The encoder reports the respective speed and angular position of the main spindle to the computer.
The computer converts this information and provides the necessary impulses for start and feed speed to the feed motor of the longitudinal slide.

Thread design

When conforming to standard metric ISO threads a separate radius of curvature (R1) is assigned to each thread pitch.
This means that for each pitch a separate threading tool is needed.
Solution:
On this machine we use a threading tool with a radius R2 = 0,04 mm for all pitches from 0,5 to 2 mm.

Disadvantage:
The radius of curvature R2 does not conform to standard. The thread turns become a little deeper, thus resulting in a smaller core diameter (see tables) than with standard threads. These threads naturally can be screwed with standard screws and nuts.

3. Design of a thread

During the design of a thread mind the following items:

Thread dimensions

The threads cut in accordance with this table can be screwed with standardized screws and nuts.

Metric threads (dimensions in mm)

Thread designation	Pitch P	Bolt		Nut	
		Nominal diameter D_1	Thread height H_1	Core diameter D_2	Thread height H_2
M3	0,5	3,00	0,337	2,459	0,285
M3,5	0,6	3,50	0,416	2,850	0,355
M4	0,7	4,00	0,490	3,242	0,414
M4,5	0,75	4,50	0,529	3,688	0,448
M5	0,8	5,00	0,551	4,134	0,479
M6	1,0	6,00	0,717	4,917	0,609
M8	1,25	8,00	0,907	6,647	0,771
M10	1,5	10,00	1,100	8,376	0,934
M12	1,75	12,00	1,285	10,106	1,098
M14	2,0			11,835	1,257
M16	2,0			13,835	1,257

* M.. stands for metric standard threads

Inch threads according to US-standard (dimensions in inch)

Thread designation	Turns per inch	Pitch P	Bolt		Nut	
			Nominal diameter D_1	Thread height H_1	Core diameter D_2	Thread height H_2
.112 (4)	40	0,0250	0,1120	0,0174	0,0813	0,0147
.125 (5)	40	0,0250	0,1250	0,0174	0,0943	0,0147
.138 (6)	32	0,0313	0,1380	0,0243	0,0997	0,0188
.164 (8)	32	0,0313	0,1640	0,0243	0,1257	0,0188
.190 (10)	24	0,0417	0,1900	0,0330	0,1389	0,0252
.216 (12)	24	0,0417	0,2160	0,0330	0,1649	0,0252
1/4	20	0,0500	0,2500	0,0386	0,1887	0,0309
5/16	18	0,0556	0,3125	0,0447	0,2443	0,0346
3/8	16	0,0625	0,3750	0,0502	0,2983	0,0391
7/16	14	0,0714	0,4375	0,0577	0,3499	0,0449
1/2	13	0,0769			0,4056	0,0485
9/16	12	0,0833			0,4603	0,0526
5/8	11	0,0909			0,5135	0,0576

1" = 25,4 mm

Chamfer

Start and end of a thread are usually provided with a chamfer of 45° or 60° to avoid a burr during thread-cutting.
The 30° chamfer can still be turned with the right lateral tool and saves an additional changing of the turning tool.

Thread lead-in - Thread lead-out

At the start and end of a thread-cutting procedure the slides (with threading tool) have to be accelerated or decelerated.

This acceleration path (approx. 3 mm) and decleration path (approx. 1 mm) has no constant thread pitch and therefore must be outside of the cutting process. The necessary free space for the turning tool has to be taken into account during the design of the workpiece.

Design of core diameter for bolts

- Draw one straight line between starting point (S) and final point (E) with the correct thread height H1 (see table)
- Search intersecting point (X) and delete projecting line

Design of nominal diameter for nut

Similar to description for core diameter of bolt.

4. Machining data

Tool change

Select tool for thread-cutting (see menu program - F3/change tool - F6).

Right-hand thread - left-hand thread

If the tool is at the right side of the thread (A) a right-hand thread is cut. If you move the tool (in rapid motion) to the left side of the thread (B) a left-hand thread is cut.

Call-up thread cycle

Select the key F3 in the menu program and the submenu CYCLES appears.
Select the thread cycle with F4.

CYCLES	
F1	ROUGHING
F2	BACK POCKET
F3	CONTOUR
F4	CUT-IN
F5	THREAD
F6	
F7	
F8	
F9	
F10	

F3 ←

PROGRAM	
F1	RAPID
F2	FEED
F3	CYCLES
F4	EDITOR
F5	TOOL CHNGE
F6	ERASE LAST
F7	ERASE PROG
F8	FEED
F9	G94/G95
F10	OFFSET

Enter number of machine cuts

After selecting the thread cycle the following message appears at the screen:

```
Enter number of machine cuts (10)
(ENTER = accept):
```

The recommended number of machine cuts is 10 to 20 according to thread height (H).
The software adjusts the feed depth so that at each cut the chip cross-section has the same size.
The feeds are carried out only in X-direction.

Thread lead-in - thread lead-out

Screen message:

```
Enter thread lead-in = 3.000, lead-out = 1.000
(ENTER = no change)
```

For thread lead in a length of at least 3 mm should be entered because the slide (with thread tool) needs an acceleration path. For thread lead-out enter approx. 1 mm. At left-hand threads these values must be provided with a negative sign.

Right-hand thread

Acknowledge inputs

End of input (ENTER = accepted)

You have two possibilities:

↵ The thread cycle is worked off on the screen.

Any key Exit from the thread cycle. The values entered last are suggested at the next call-up of the thread cycle.

Thread pitch

Screen message:

Pitch = 1.000 (ENTER = no change):

Thread-cutting tools offered are designed for pitches of P = 0,5 - 2 mm or 11-40 turns per inch. You can see the pitches for standardized threads from the table of the chapter "thread dimensions".

Indicate nominal and core diameter

Enter nominal diameter element:

Place the cursor to the nominal diameter and acknowledge with ENTER. The following message appears:

Enter core diameter element:

Place the cursor to the core diameter and acknowledge with ENTER.

Notes for working off the thread cycle at the machine

Speed:
Adjust the smallest speed range (BC1 = 200 rpm).

Example: Generation of a special screw

- Unmachined part: Ø22,100 with bore Ø3,5
- Drawing:

- Roughing
- Back-off
- Execute contour cycle
- Select thread tool
- Position thread-cutting tool at the right side of the thread
- Select submenu "thread"

F4

Enter number of machine cuts (10)
(ENTER = accepted):

15 ↵

Enter thread lead in = 3.000, lead out = 1.000
(ENTER = no change)

↵

Enter pitch = 1.000 (ENTER = no change):

1.25 ↵

Enter nominal diameter element:

Cursor to nominal diameter and acknowledge with ↵

Enter core diameter element:

Cursor to core diameter and acknowledge with ↵

End of input (ENTER = accepted)

↵

Important: During machining and particularly during thread-cutting the workpiece is to be supported in any case by a lathe center.

4.3.4 Edit (F4)

By pressing the key F4 you enter the "Edit"program. In Edit you can work in the same way as with the original CNC machine and set up NC programs according to standards (acc. to DIN 66025 and ISO 1056). For an exact description see chapter 6 "EDIT".

Machining possibilities

(Survey)

Workpiece data

CAD (design on screen)

EDIT
(determine machining with CNC program acc. to DIN 66025 and ISO 1056)

```
N0000/G67/F3
N0010/G94/F200/T01
N0020/G00/X20/Z1
N0030/G84/X16.1/Z-44.8 /D3 800/F180
N0040/G00/X16.1/Z1
N0050/G84/X12.1/Z-24.8 /D3 800/F180
N0060/G00/X12/Z1
N0070/G01/Z-25
N0080/G01/X16
N0090/G01/Z-45
N0100/G01/X20
N0110/G00/X40/Z20
N0120/M30
```

1 transform 2

CAM
(Determine machining process graphically and simulate)

Work off

Ref. 1: The machining process (CAM) determined graphically can be transformed into NC-programs and changed there (in the editor).

Ref. 2: The NC-program entered in the screen is transformed by pressing a key and is simulated automatically on the screen. The machining process determined in the editor can also be changed graphically again and in the following be executed on the lathe.

4.3.5 Tool change (F5)

Tool change during the compilation of the machining program
Procedure
- Move the current tool to a tool change position (with F1 = rapid)
- Press F6, a tool range appears on the screen.

- Select the tool (e.g., 3 ↵)
- The machining program to be compiled can be continued with the new tool after the tool change.
- The travelling movements of the new (active) tool are shown in light blue, those of the old tool in dark blue.

Tool change during the processing of a machining program
The machining program remains on the machine during the processing and automatically stops before the tool change on the PC (= intermediate stop).
Carry out the tool change on the PC
1. Select the menu program
2. Press F6
3. Enter the desired tool number and confirm with Enter

Carry out the tool change on the machine
4. Change the tool by hand
5. Select the menu setup and re-synchronize the machine - PC
6. Select the menu machine
7. Select automatic
8. Press enter

4.3.6 Erasing the last movement (F6)

The previously entered movement is erased, when the key F7 is pressed.

4.3.7 Erasing the machining program (F7)

The complete program is erased.

4.3.8 Entering the feed values (F8)

The set feed value for F2 (= moving with feed speed) can be altered, when the key F8 is pressed.

Screen message	Entry
	F8
Feed = 30 (ENTER = no change):	40 ↵

4.3.9 Change feed unit (F9)

A feed value can be entered in two different units:
- in mm/min (= G94)
- in mic/r (= G95)
  = 1µm/r = 1/1000 (0.001) mm/r

F (mm/min) = F (mm/r) x S (r/min)

By pressing the F9 key the feed unit on the screen changes. The adjusted nominal value remains the same.

Example:
F = 80 mm/min means: The cutting tool travels 80 mm in one minute, no matter how often the workpiece turns.

F = 80 mic/r means: The cutting tool travels 0,08 mm during one workpiece revolution.

At a speed of e.g. 200 rpm there is a feed of 0,08 mm x 200 rpm = 16 mm/min.

4.3.10 Entering the finishing offsets in X and Z (F10)

The set finishing offsets can be altered, when the key F10 is pressed.

Note
The entry of the finishing offset in the X direction refers to the radius of the work piece.

Screen message	Entry
	F10
Offset in X, Z 0.100 0.200 (ENTER = no change):	0.05, 0.1 ↵

4.4 DISPLAY (F4)

The screen display is determined or altered in this menu.

	DISPLAY
F1	ZOOM ALL
F2	ZOOM WINDW
F3	ZOOM PIECE
F4	HEAD STOCK
F5	WORK PIECE
F6	TOOL
F7	TOOL PATH
F8	TAIL STOCK
F9	CHANGE SIM
F10	

4.4.1 Zoom all (F1)

Everything shown on the screen (head stock, work piece, work piece contour, ...) is enlarged, so that it can still be displayed within the screen.

F1 ... The complete screen area is enlarged.

4.4.2 Zoom window (F2)

A "screen window" is defined and enlarged across the whole of the screen area.

Two diagonal bits are determined with the cursor keys and ENTER. The right-hand bit defined in this way is then enlarged across the whole of the screen surface.

Example

F2

Screen message

Position cursor. ENTER = starting point:

Move the cursor to the 1st bit point (X = 3 / Z = -4 mm).

Move cursor. ENTER = accept position:

Move the cursor to the 2nd diagonal bit point (X = 10 / Z = 17mm).

4.4.3 Zoom work piece (F3)

The work piece shown on the screen is enlarged, so that it is still shown within the screen, when the key F3 is pressed.

4.4.4 Removing, inserting the head stock (F4)

The head stock shown on the screen can be removed with the key F4.

The head stock is re-inserted, when F4 is pressed again.

4.4.5 Removing, inserting the work piece (F5)

The work piece shown on the screen can be removed with the key F5.

The work piece can be re-inserted, when F5 is pressed again.

4.4.6 Removing, inserting the tool (F6)

The tool shown on the screen can be removed with the key F6.

The tool is re-inserted, when F6 is pressed again.

4.4.7 Removing, inserting the tool path (F7)

The tool path shown on the screen can be removed with the key F7.

The tool path is re-inserted, when F7 is pressed again.

4.4.8 Inserting, removing the tail stock (F8)

The tail stock can be inserted on the screen with the key F8.

The tail stock is removed again, when F8 is pressed again.

4.4.9 Changing the simulation display (F9)

You can select two simulation displays with the key F9.

1. Displaying the tool path
   (= basic setting)

2. Simulating the cutting

4.4.10 Hotkeys - machine

You can move the slides and influence the machining cycles by means of these keys.

↑	Move the slides in the -X direction
↓	Move the slides in the + X direction
←	Move the slides in the -Z direction
→	Move the slides in the + Z direction
PgUp↑	Enlarge the travel path of the stepping motors
PgDn↓	Reduce the travel path of the stepping motors
>	Increase the feed
<	Reduce the feed
Space bar	Stop the slides (= intermediate stop)
Esc	Interrupt the machining

Travel path of the stepping motors

PgUp↑	10 mm (= largest travel path)
PgUp↑	5 mm
	1 mm (= switch-on condition)
PgDn↓	0,5 mm
PgDn↓	0,1 mm
PgDn↓	0,05 mm
PgDn↓	0,01 mm (= smallest travel path)

Altering the programmed feed

>	200 %
>	150 %
>	120 %
	100 % (= switch-on condition)
<	80 %
<	60 %
<	50 %
<	40 %
<	30 %
<	20 %
<	10 %
<	5 %
<	0 %

4.5 Set up 0,0 (F5)

The measuring system of the machine and the PC must be synchronized with one another. This is carried out with the "scratching" (on the diameter and the length) of the clamped work piece.

Note
You must synchronize the machine and PC, before you select the menu points MANUAL or MACHINE.

Procedure
- Press F5
- Switch on the main spindle
- Move the tool with the arrow keys (see hotkeys - machine) to the diameter of the clamped work piece (in rapid) and "scratch" the work piece.

Note
In order to position the work piece exactly, you must reduce the step size with the PgDn key.

- After reaching the desired diameter, press the enter key.
- Enter the X value (diameter values) of the work piece and confirm it.
- Move the tool with the arrow keys to the face of the clamped work piece and "scratch" the face of the work piece.

Note
In order to position the tool exactly, you must reduce the step size with the PgDn key.

- After reaching the desired length, press enter.
- Enter the Z value (length) of the work piece.

- The measuring system of the machine and PC is now synchronized. You can process your compiled machining program on the machine.

Example
Workpiece Ø 20 mm, length = 100 mm

F5

Screen message

Tool with ↑ ↓ to the diameter aligning pos. Step = 1,00 (ENTER = end):

Move the tool to the diameter and reduce the step size with PgDn, until the tool is positioned exactly on the aligning diameter.

Input diameter for this tool position:

20

Tool with ↑ ↓ to the length aligning pos. Step = 1,00 (ENTER = end)

Move the tool to the aligning length and reduce the step size with PgDn, until the tool is positioned exactly on the length of the work piece.

Input Z value for this tool position

100

4.6 Manual (F6)

Moving the machine in "manual mode". You can move the tool in rapid on the machine and screen at the same time by means of the arrow keys (see hotkeys - machine).

Condition
The machine was aligning (menu aligning).

Procedure

F6

↓

Screen message

Move the tools with the cursor keys, step = 1.00:

↓

Move the tool. The step size can be altered with PgDn ↓ or PgUp ↑ .

The respective position of the tool is displayed on the screen.

4.7 Machine (F7)

Processing the compiled machining program on the machine

Condition
The machine was aligning (menu aligning).

MACHINE	
F1	AUTOMATIC
F2	SINGLE
F3	FAST RUN
F4	EMPTY CUT
F5	REPEAT
F6	CURSOR
F7	SHIFT PROG
F8	SCALE PROG
F9	
F10	

4.7.1 Automatic (F1)

The machining program compiled on the program menu is processed automatically.

Affecting the machining

[>]	.... Increasing the feed
[<]	.... Reducing the feed
[Space bar]	.... Intermediate stop, the slides stop
[Esc]	.... Interruption of the machining

4.7.2 Single (F2)

The machining program is processed in blocks. Every program block must be confirmed with the enter key before the processing.

You exit from this menu point with the Esc. key.

4.7.3 Fast run (F3)

The complete machining program is automatically processed in the rapid.

Application
Testing a machining program without a work piece

4.7.4 Empty cut step (F4)

Move to the desired machining step with the arrow key. The machining program can be started from this block.

Procedure

[F4]
↓
Screen message

Position at desired program steps with ↓ ↑ keys (ENTER = end):

Possibilities

[POS 1]	.... Searching block by block from the start of the program
[END]	.... Searching block by block from the end of the program
[↓]	..... Return block by block
[↑]	..... Forwards block by block
[↵]	..... Confirm the desired block

The machining program can be started from this desired block.
(Automatic, single or fast run)

Application
Re-machining of certain contour parts

E.g., finishing the fit on a tool with overmeasure. Measure the fit on the clamped tool after the machining.
Move to the fit with "F4 empty cut step".
Process the required differential measure in Single in the X direction with "F7 shift program".

4.7.5 Repeating the program (F5)

Unit: [mm]

A machining program can be shifted in the X and Z direction (X = radius dimension) and be repeated. The number of the steps determines how often the machining program is to be shifted and repeated.

Application
Finishing several, similar parts from one work piece

Procedure

4.7.6 Moving the tool to the cursor (F6)

The tool is moved from its present position to the cursor position with the set feed, whereby the machine moves along the same movement.

This movement is not copied into the machining program.

4.7.7 Shifting the program (F7)

Unit: [mm]

A machining program can be shifted in the X and Z direction (X = radius measurement).

Application
Re-turning the work piece contour

4.7.8 Altering the scale program (F8)

A machining program can be enlarged or reduced. The altered scale is assumed as the new scale M1:1 after the enlargement or reduction.

Examples:
Reduction
Input: 0.5 = M1:2

Enlargement
Input: 2 = M2:1

Procedure

Attention!
Observe the size of your work piece, if you alter the machining program in the scale.

4.8 Archive (F8)

The programs compiled by you are stored or recalled in this menu.

Drawings created by you or machining programs must be stored as files before leaving the program.

4.8.1 Storing the program (PRG) (F1)

Screen message	Entry
	F1
Input file name to be stored:	demo1 ↵

The machining program is stored under the name "demo1".

In the sub directory (e.g. COMPACT) a program is stored in two files:

DEMO1.GEO
DEMO1.NCP

DEMO1 is the name you have choosen (max. 8 signs) and an automatic addition (.GEO resp. NCP).

.GEO in this file there are the informations about workpiece, drawing (CAD) and tool path (CAM).

.NCP in this file the NC-program is stored.

4.8.2 Loading the program (PRG) (F2)

A stored machining program can be recalled.

Example 1: Call-up of a program

Screen message	Entry
	F2
Input file name to be retrieved:	demo1 ↵

The machining program "demo 1" is loaded.

Example 2: Listing all stored programs

All stored programs are displayed in the area menu display, when the sign * is entered.

Screen message	Entry
	F2
Input file name to be retrieved:	* ↵

The stored programs are displayed and can be selected by means of the ↓ ↑ keys and recalled with ↵ .

4.8.3 Storing the geometry (GEO) (F3)

A work piece drawing can be stored.

Screen message	Entry
	F3
Input file name to be stored:	part 1 ↵

The work piece drawing is stored under the name "part 1".

The work piece geometries (are stored under the name entered by you in your sub-directory (e.g.: COMPACT) and are automatically provided with the suffix **.DXF**.
The suffix **.DXF** means that the work piece geometries are stored in the DFX format. I.e.: you can read AUTOCAD drawings in the DXF format into this software or work piece geometries created with this software can be copied into the AUTOCAD software.

4.8.4 Loading the geometry (GEO) (F4)

A stored work piece drawing can be recalled.

Screen message	Entry
	F4
Input file name to be retrieved:	part1 ↵

The work piece drawing "part1" is loaded.

Listing all stored work piece drawings works as with programs (chapter 4.8.2).

Erasing the stored programs and geometries

- Exit the software (remain in the sub-directory COMPACT).
- All files in the sub-directory COMPACT are listed by means of the DOS command "DIR".
- Erasing the respective file (see DOS manual).

4.9 Print (F9)

The respective screen display is printed.

Condition
The printer is connected, switched on and set in the IBM graphics mode (see printer manual).

Screen message	Entry
	F9
Is the printer connected and on "line"? (ENTER = yes)	↵

Possibilities
ENTER = printing
Any other key = interruption

4.10 End (F10)

Exit from the program

Attention!
You must store the programs compiled by you beforehand (archive menu).

Screen message	Entry
	F10
Are you sure? (ENTER = yes):	↵

Possibilities
ENTER = exit from the program
Any other key = remaining in the program

The title screen appears again:

Esc . . . Return to DOS

5. SPECIMEN EXAMPLES

Chess figures

These specimen examples serve to re-inforce the knowledge acquired concerning the programming of this EMCO software.

The creation of the contour of these examples is explained step by step on the following pages.

We recommend you to use aluminium and brass with a diameter of 20 and a length of 50-70 mm for the chess figures due to chip removal reasons.

You will find the technological data, like
- max. main spindle speeds
- max. feed
- max. cutting depths

in chapter A "machine description".

The figures are designed in such a way that the back-off cycles can be worked off with the right lateral steel.

When preparing the program it is advantageous to execute the roughing cycle at the extreme right as last processing cycle. Thus workpieces can be supported with the lathe center until the last working cycle.

The lathe center has to be removed in time in the last working cycle to avoid a collision with the tool.

Pawn

1. Creating a work piece
(See work piece (F1))
Ø = 20 mm, length = 52 mm, center hole = 4,5

Drawing aid
The work piece is enlarged across the whole of the screen surface by means of the hotkey [Z] .

2. Drawing the work piece contour
(See submenu point, line/hotkeys)
Note
Only the lower half of the work piece contour must be drawn.

Contour points	X direction	Z direction
1	0	-5
2	0	-11
3	6	-11
4	3	-22
5	3	-27
6	6	-27
7	10	-34
8	10	-37

3. Adding the radii and chamfers

4. Cleaning

5. Splitting the arc (F4)

6. Adding the visible edges (F5)

7. Mirroring the work piece contour (F6)

8. Storing the work piece contour
(See ARCHIVE (F8))

Bishop

1. Creating a work piece
(See work piece (F1))
Ø = 20 mm, length = 85 mm, center hole = 4,5

Drawing aid
The work piece is enlarged across the whole of the screen surface by means of the hotkey [Z].

2. Drawing the work piece contour
(See submenu point, line/hotkeys)
Note
Only the lower half of the work piece contour must be drawn.

Contour points	X direction	Z direction
1	-5	-1
2	-5	-13,5
3	-5	-26
4	4	-15
5	6	-25
6	4	-35
7	4	-42
8	6	-45
9	8	-45
10	8	-48
11	5	-55
12	5	-59
13	7	-59
14	10	-65

3. Cleaning

4. Adding the radii and chamfers

5. Splitting the arc (F4)

6. Adding the visible edges (F5)

7. Mirroring the work piece contour (F6)

8. Storing the work piece contour
(See ARCHIVE (F8))

Queen

1. Creating a work piece
(See work piece (F1))
Ø = 20 mm, length = 87 mm, center hole = 4,5

Drawing aid
The work piece is enlarged across the whole of the screen surface by means of the hotkey [Z] .

2. Drawing the work piece contour
(See submenu point, line/hotkeys)
Note
Only the lower half of the work piece contour must be drawn.

Contour points	X direction	Z direction
1	0	-5
2	0	-10
3	5	-10
4	2	-8
5	7	-8
6	7	-10
7	5	-15
8	5	-20
9	10	-20
10	10	-22
11	8	-25
12	10	-30
13	3	-47
14	3	-53
15	5	-53
16	5	-55
17	10	-60
18	8	-66
19	10	-72

3. Cleanimg

4. Adding the radii and chamfers

5. Adding the visible edges (F5)

6. Mirroring the work piece contour (F6)

7. Storing the work piece contour
(See ARCHIVE (F8))

King

1. Creating a work piece
(See work piece (F1))
Ø = 20 mm, length = 90 mm, center hole = 4,5

Drawing aid
The work piece is enlarged across the whole of the screen surface by means of the hotkey [Z].

2. Drawing the work piece contour
(See submenu point, line/hotkeys)
Note
Only the lower half of the work piece contour must be drawn.

Contour points	X direction	Z direction
1	0	-3
2	4,5	-3
3	4,5	-4,5
4	3	-8
5	7	-8
6	7	-10
7	5	-18
8	10	-22
9	10	-25
10	7	-50
11	7	-58
12	10	-58
13	10	-60
14	8	-66
15	10	-72

3. Adding the radii and chamfers

4. Adding the visible edges (F5)

5. Mirroring the work piece contour (F6)

6. Storing the work piece contour
(See ARCHIVE (F8))

Rook

1. Creating a work piece
(See page work piece (F1))
Ø = 20 mm, length = 65 mm, center hole 8.5, 5

Drawing aid
The work piece is enlarged across the whole of the screen surface by means of the hotkey Z .

2. Drawing the work piece contour
(See submenu point, line/hotkeys)
Note
Only the lower half of the work piece contour must be drawn.

Contour points	X direction	Z direction
1	0	-4
2	7	-4
3	7	0
4	9	0
5	9	-16
6	8	-19
7	6	-44
8	10	-44
9	10	-50

3. Adding the radii and chamfers

4. Adding the visible edges (F5)

5. Mirroring the work piece contour (F6)

6. Storing the work piece contour
(See ARCHIVE (F8))

Note on inside turning

- Since inside turning is carried out in over-mounted form (without center punch support), feed (F) and advance should be as small as possible.
  F = approx. 20 mm/min
  Advance approx. 0,2 mm

- For inside turning of small diameters the inside cutting tool has to be set obliquely to avoid collisions (A). However, mind in any case that a clearance angle remains behind the tool tip to guarantee a perfect cut.

Knight

1. Creating a work piece
(See work piece (F1))
Ø = 20 mm, length = 80 mm, center hole = 4,5

Drawing aid
The work piece is enlarged across the whole of the screen surface by means of the hotkey [Z].

2. Drawing the work piece contour
(See submenu point, line/hotkeys)
Note
Only the lower half of the work piece contour must be drawn.

Contour points	X direction	Z direction
1	0	-5
2	7	-5
3	7	-17
4	4	-35
5	4	-38
6	3	-40
7	3	-45
8	4	-47
9	4	-49
10	10	-54
11	8	-58
12	10	-62
13	10	-65

3. Adding the radii and chamfers

4. Adding the visible edges (F5)

5. Mirroring the work piece contour (F6)

6. Storing the work piece contour
(See ARCHIVE (F8))

Extension for the knight

Working with the vertical equipment

Tool: roughing mill Ø 8 mm

6. EDIT

Introduction

With the COMPACT 5 PC software you have not only an efficient CAD/CAM software, but you can also work like at a normal CNC machine and set up standard (DIN 66025 and ISO 1056) NC programs.

Explanation of term EDIT
The procedure of entering data (numbers and letters) into a PC or in a control is also called edit.

Example:
An NC program is entered into the computer (edited). This is carried out in a certain part of the COMPACT 5 PC software, the EDIT.
EDIT = mode for program input

What is programming
Programming = communicating data in a settled sequence and language which the computer can understand and transform. The NC-programming language and the NC-program structure was determined by experts in DIN 66025 and ISO 1056.

Setting up an NC-program
When you are setting up an NC-program you have to give exactly the same information and instructions to the computer as are needed by someone for machining a workpiece on a hand-operating machine who has no idea of turning.

This information is:

1. Geometrical data
   You find these geometrical data (dimensions) on the drawing of the workpiece.
2. Technological data
   Feed, spindle speeds, tools, etc.
3. Machine functions
   These are described in this chapter (see G, M-functions)
4. Program instructions
   When setting up an NC program certain instructions on sequences of commands (see G, M-functions) have to be adhered to.

Geometrical Data

Coordinate system

The information "move longitudinal slide in direction headstock" is a very long one. Besides that, in each language it would be different. That's why the traverse path movements with machine tools are described within the coordinate system.

Lettering of drawing

The geometrical information is in the technical drawing. Drawings can be done according to the incremental or to the absolute system. In many cases you find a combination of these two systems: incremental and absolute.

Absolute system
There is one point of reference.

Incremental system
Each measurement is based on the previous one.

Combined system
The measurements 1 and 2 are absolute ones, i.e. based on one point of reference. The measurements 3 and 4 are incremental ones.

Methods of programming

In the program you have describe in each block the path of the turning tool. Basically there are two methods to describe this path.

1. Absolute value programming:
The points which the turning tool has to approach are indicated by a zero point with X and Z. The X value is a diameter value.

Pt.	X	Z
1	6	0
2	6	-2,5
3	4	-2,5
4	4	-4
5	0	-6

2. Incremental dimension programming:
Here the incremental dimensions are entered with U and W. The initial point for each path description is the actual position of the turning tool. The U value is a radius value.

Pt.	U	W
1	-3	0
2	0	-2,5
3	1	0
4	0	-1,5
5	2	-2

The zero point of the workpiece = reference point

The reference point (R) is on the Z axis at the right blank edge.

Attention:
A new blank has to be set up before editing an NC-program.
Reason:
Otherwise all the values entered into the NC program do not refer to the reference point (R) but to the machine zero point (M).

Technological Data

The feed (F)

The feed can be programmed in mm/min or in mic/r (= µm/r). For further information see chapter G94/G95.

Tool programming (T)

The call-up of tools and the tool change are programmed with the letter T and a two-digit number

e.g. T01

The T is the abbreviation of the English word "Tool".
The two-digit number indicates the tool number.

Tool number:

T01	right side tool
T02	neutral side tool
T03	left side tool
T04	parting-off tool
T05	cut-in tool for locking rings
T06	left thread-cutting tool
T07	right thread-cutting tool
T08	inside boring tool
T09	inside cut-in tool for locking rings
T10	right inside thread-cutting tool
T11	left inside thread-cutting tool
T12	not occupied

Note
A G00 (rapid motion) has to be programmed as next travel motion after each new tool call-up.

Possibility 1: in the same block
N ..../T02/G00/X .... /Z ....
Possibility 2: in the next block
N ..../T02
N ..../G00/X ... /Z ....

Program Structure

Parts of the NC program

1. The program

All essential data for the manufacture of a work piece are filled in.
The structure of such a program is standardized.

```
N0000/G67/F3
N0010/G94/F150/T01
N0020/G00/X25/Z2
N0030/G01/X21/Z1/F100
N0040/G84/X16/Z-10/D3 = 500/F100
N0050/G00/X35/Z10
N0060/G56
N0070/M30
```

2. The block

The program consists of blocks. A block contains all data necessary to execute an operation.

3. The word

Each block consists of various words. Each word has a special meaning.

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

The words of the Compact 5 PC

N:
N indicates the block number. Practicable block numbers are N0000 up to N9999. It is appropriate to number the program blocks in steps of tens.

G:
G is a symbol for the path information. Each figure is allocated to a certain kind of motion. e.g. 01 = move in straight line 03 move in circle, etc.

X:
X means indication of the absolute position of the X axis. The numerical value may have the operational sign " + " or " - " (X = diameter value).
Input: mm

Z:
Z means absolute indication of the position of the Z axis. The numerical value may have the operational sign " + " or " - ".
Input: mm

U:
U means incremental indication of the position of the X axis. The numerical value may have the operational sign " + " or " - " (U = radius value).
Input: mm

W:
W means incremental indication of the position of the Z axis. The numerical value may have the operational sign " + " or " - ".
Input: mm
Note: The addresses X, Z, U, W can also be programmed mixed in one block.

I:
I means an incremental path indication in direction X with circle movements (G02/G03). I is an auxiliary indication for the calculation of the circular traverse motion. The numerical value may have the operational sign " + " or " - ".
Input: mm

K:
K means an incremental path indication in direction Z at with circle movements (G02/G03). K is an auxiliary indication for the calculation of the circular traverse motion. The numerical value may have the operational sign " + " or " - ".
Input: mm

F:
Under the F address different information can be programmed:
1. F in connection with G94:
   Under the F-address the feed is programmed in mm/min.
2. F in connection with G95:
   Under the F-address the feed is programmed in µm/min.
3. F in connection with G85:
   Under the F-address the thread pitch is programmed in µm/min.
4. F in connection with G67:
   Under the F-address the colour of the tool path is determined (for graphic simulation).

T:
With T a certain tool can be called up. Possible tool addresses are T01 up to T11.

M:
M is the abbreviation of the English word "miscellaneous". Switching and additional functions are determined with M.
Note: On the Compact 5 PC only M30-program end can be programmed.
On industrial machines functions such as e.g. "COOLANT ON/OFF", CLAMPING DEVICE OPEN/CLOSE" are programmed with M-functions.

The parameters of the Compact 5 PC

With the parameters P and D additional information for working off the cycles (G84, G85, G86) are communicated to the PC.

e.g. with G84 a cone can be programmed with P_0 in ± X direction.

The D-parameters must be entered in each case.
The P-parameters have to be programmed only if the value is not 0.

Survey of G and M functions

G00	Rapid motion
G01	Feed motion (straight line interpolation)
G02	Circle interpolation in clockwise direction
G03	Circle interpolation in counterclockwise direction
G56	Deselection of reference point offset
G59	Call-up of reference point offset
G67	Indicate colour of tool path
G84	Longitudial turning and facing cycle
G85	Thread cycle
G86	Cut-in cycle
G92	Reference point offset
G94	Feed speed in mm/min
G95	Feed speed in µm/r
M30	Program end

Specifications on the sequence of words:

Apart from the X(U), Z(W) sequence in the cycle G84 there is no absolute rule on the word sequence. However, so as to obtain a clear program structure, you should observe the following sequence:

* Each block starts with the block number N.
* The G-function should be programmed after the block number.
* Words for the coordinates X(U), Z(W).
  Observe the reversal of the X(U), Z(W) sequence in the cycle G84.
* Where G02, G03 is programmed, the interpolation parameter I, K should be programmed after X(U), Z(W).
* Where cycles are programmed, the parameters should be programmed after the X(U), Z(W) addresses.
* The F-word (feed thread pitch).
* The T-word (tool address).
* The M-word (additional functions).

Self-holding functions, words

* G and M functions and also other words are self-holding. That means that they are active until they are overwritten or selected.

  Example:
  N100/...
  N110/G00/X50.000/Z20.000
  N120/X40.000/Z18.000
  N130/G01/X30.000/Z15.000/F100
  N140/...

  Explanation:
  G00 is also effective in block 120 and is overwritten with G01 in block 130.

* X(U), Z(W), F, T word contents are taken over in the following blocks:

  Example:
  N200/...
  N210/G01/X40.000/Z25.000/F120
  N220/Z12.000
  N230/G00/X35.000/Z32.000
  N240/...

  Explanation:
  In block 220 G01, X = 40 and F120 is effective. F120 remains effective until another value is programmed under the F address.

G-Codes, their formats and description of formats

Specific addresses are asigned to most G-codes.

Example:
G00/X ± ....,.../Z ± ....,...
or
G01/X ± ....,.../Z ± ....,.../F....
For a short and easy to understand description of pertaining addresses (format description) the data are encoded.

Code:
1) Instead of giving the possible inputs, the number of decades is given.

   Example:
   Instead: N from 0 to 9999
   or N.... we write N4.
   $N\underbrace{....}_{4} \rightarrow N4$

2) The specification of the possible decades before or after a decimal point is coded with two figures.
   $X\underbrace{....}_{4}\underbrace{,...}_{3} \rightarrow X43$
   The first figure: Decade before decimal point
   The second figure: Decade after decimal point

3) If the values could be negative or positive a + sign is written between address and number.
   X + 43
   Remark: For better determination quite often a ± sign is written (X ± 43). Underneath the input format there are the different dimensions shown.

   Example:

N4	G01	X / U ±43	Z / W ±43	F4
		[mm]	[mm]	[µm/r] [mm/min]

N4: four digits without decimal point and sign

X, Z, U, W: ± sign possible, four digits before decimal point, three digits after decimal point

F4: four digits without decimal point and sign

Guidelines for working in EDIT

On the following pages you can find the description of the functions with which you are able to set up NC programs.
The G functions are arranged in rising order (G00 up to G95). If you are learning this kind of programming you should mind the following:

1. Guideline for program start and program end:

Program start:
G67/F → determine colour of tool path
G94(G95) → feed in mm/min or µm/r
F → feed programming
T → tool programming
G00 → rapid motion
After each tool call-up the 1st traverse movement must be a G00 block.

Program end:
M30 means end of program. M30 don't have to be programmed at this software.

2. Sequence for beginners for learning how to program:

G67	determine colour of the tool path
T	tool programming
F	feed programming
G94/G95	feed in µm/r, mm/min
G00	rapid motion
G01	feed movement
G84	longitudinal turning cycle
G84	facing cycle
G86	cut-in cycle
G85	threading cycle
G02/G03	circular interpolation
G92/G59 G56	offset position

Also study all the exercises for the individual G-functions. These are set up in such a way that you are introduced to programming step by step.

For the G-functions there is a specific summary. This is framed in grey and serves also as a short description.

G00 - Rapid mction

Input format:

N 4	G00	X ± 43 / U	Z ± 43 / W
		[mm]	[mm]

N ..... Block number
G00 .. Rapid motion
X(U) } Absolute (incremental) coordinates
Z(W) } of the target point Z

The positioning of the turning tools, i.e. movement of same without chip removal, must be done with highest possible speeds (rapid motion) for economic reasons.
The slides move simultaneously in X and Z direction.

Rapid motion speed: PC/AT: 700 mm/min
PC/XT: 350 mm/min

G00 (rapid motion) is merely a traversing movement - no working movement!

Notes:
- It does not matter in which sequence X (U), Z (W) are written.
- They can also be programmed mixed in one block (absolute and incremental).
  e.g.: G00/X44.000/W-9.000

Example:

absolute: N.../G00/X30/Z2
incremental: N.../G00/U-6/W-9

G01 - Straight line interpolation

Input format:

N 4	G01	Z / W ± 43	F 4
		[mm]	[µm/r] [mm/min]

N ..... Block number
G01 ... Straight line interpolation
X(U) } Absolute (incremental) coordi-
Z(W) } nates of the target point Z
F ..... Feed

G01 is a linear operating movement. The feed must be programmed. It can be input in [mm/min] (G94) or in [µm/r] (G95).
The feed (F) is self-holding.
X = diameter value
U = radius value

Possibilities of G01:

1. Turning in Z direction (longitudinal turning)
Longitudinal turning with defined feed speed. The X movement is zero. No value is entered for X (U).

N.../G01/-Z (-W)/F...

2. Turning in direction X (facing)
Facing with defined feed speed.
The Z movement is zero. No value is entered for Z(W).

N.../G01/X(U)/F...

3. Turning in X and Z direction (taper turning)
With taper turning longitudinal and lateral slides must move simultaneously. In accordance with the taper angle the slides have to traverse at a certain ratio. The computer (PC) calculates a lot of intermediate values referring to this straight line and transmits the respective traverse commands to the slides.
This finding of intermediate values of a straight line is called straight line interpolation.
The straight line can have any angular position.
With longitudinal turning and facing no interpolation takes place.

Example:

absolute:
N100/.....
N110/G00/X22/Z2
N120/G00/X18
N130/G01/Z-50/F...
N140/G01/X20/Z-52/F...
N150/G00/X22/Z2
N160/.....

incremental:
N100/.....
N110/G00/.....
N120/G00/U-2 (Radius!)
N130/G01/W-52/F...
N140/G01/U1/W-2/F
N150/G00/U1/W54
N160.....

G02/G03 - Arc interpolation

Block format:

N 4	G02 G03	X U ± 43	Z W ± 43	I ± 43	K ± 43	F 4
		[mm]	[mm]	[mm]	[mm]	[µm/r] [mm/r, in]

N ..... Block number
G02... Arc interpolation clockwise
G03... Arc interpolation counter-clockwise
X (U) } Absolute (incremental) coordinates
Z (W) } of the target point
I, K .... Arc centre point coordinates
F.... Feed

1. General

- With circle turning longitudinal and lateral slide must move at the same time. In contrast to the straight line interpolation the ratio X:Z is changing continuously. The computer (PC) calculates a lot of intermediate values referring to the circular arc and transmits the respective traversing commands to the slides. This finding of intermediate values of a circle is called circle interpolation.
- In one block, arcs up to a maximum of 180° can be programmed.

2. G02/G03

Rotation G02 clockwise — Rotation G03 counter-clockwise

3. Target point coordinates X,Z/U,W

With X,Z the target point (Z) of the circular arc is described (absolute) from the reference point (R). X is a diameter value.

With U, W the target point (Z) of the circular arc is described from the starting point (S) (incremental). U is a radius value.

4. Coordinates of the center of the circle I,K

The computer knows the starting point (S) and the target point (Z) but not yet the value of the radius. Therefore the coordinates of the center of the circle are described (I,K).

Imagine the I, K system of coordinates in the center of the circular arc and describe from there the starting point of the circular arc.

The coordinates I, K are programmed always incremental.

Example 1

absolute: N.../G03/X20/Z-35/I -5/K 0 /F...
incremental: N.../G03/U5/W-5/I -5/K 0/F...

Example 2

absolute: N.../G02/X20/Z-43/I 0/K 3/F...
incremental: N.../G02/U3/W-3/I 0/K 3/F...

Example 3

absolute: N.../G03/Z-43.33/I -20/K 9.16/F...
incremental: N.../G03/W-18.33/I -20/K 9.16/F...

G56 Deactivation of the reference point offset

Input format:

N4	G56

Explanation see G92 "The reference point".

G59 Activation of the reference point offset

Input format:

N4	G59

Explanation see G92 "The reference point".

G67 Colour of the tool path

Input format:

N4	G67	F2

F....Colour code see table

For the simulation of your NC-program it is possible to change the colour of the tool path with G67.

The block with G67 has to be programmed before that block which describes the tool path.
G67 is self-holding and can be used in the program as often you want.

When using a black-and-white screen (Herkules board) the function G67 is of no significance.

Series 1: dark	Series 2: clear
F0 black	F8 dark grey
F1 dark blue	F9 light blue
F2 dark green	F10 light green
F3 turquoise	F11 light blue
F4 dark red	F12 light red
F5 lilac	F13 violet
F6 orange	F14 yellow
F7 grey	F15 white

The colours can be a little different on each screen.

Example:

```
N....   G67/F6
N....   G00/X..      ← orange tool path
N....   G67/F14
N....   G84/X../Z..  }
          :   :      }  yellow tool path
```

G84 - Longitudinal turning cycle

Input format:

N 4	G84	X U ± 43	Z W ± 43	P_0 P_2 ± 43	D_3 5	F 4
		[mm]	[mm]	[mm]	[µm]	[µm/r] [mm/min]

N ..... Block number
G84 .. Longitudinal turning cycle
X(U) } Absolute (incremental) coordinates
Z(W) } of the contour corner point
P_0 ..... Taper dimension in X (radius value)
P_2 ..... Taper dimension in Z
D_3 .... Cut division
F ..... Feed

If there is no P_0 or P_2, it need not be programmed.

Cycle definition
A cycle is a movement sequence of the tool which consists of a series of individual movements.

⇨ G00 Movement with rapid motion

→ G01 Movement with programmed feed speed (F) in mm/min or µm/r.

A cycle always has a closed movement sequence: starting point (S) = final point (E)
The starting point (S) is approached in the block before G84.

Contour corner point (K)
The movement sequence of a cycle is defined by starting point (S) and contour corner point (K). In accordance with the position of S and K to each other there are four kinds of cycle that you get as a result.
K is described with X,Z absolute or with U,W starting from S (incremental).

Cut distribution (D_3)
Frequently, a cyle is worked off in several cuts. This is indicated with cut distrubtion D_3. The movement sequence consists of the steps 1 to 12.

Taper overmeasure (P_0,P_2)
A cycle can be produced with an additional taper overmeasure in X-direction (P_0) and (or) in Z-direction (P_2).

The taper overmeasures P_0, P_2 are programmed always incremental.

The input for taper overmeasure P_0, P_2 must be carried out in the feed direction. For the example drawn below P_0 and P_2 must be a negative value, otherwise alarm 31 is effected.

To obtain the same taper, a larger value has to be inserted if starting point S is before the workpiece edge (right).

Example 1
Longitudinal turning cycle without cut distribution

absolute: N..../G84/X18/Z-40/D_3 1000/F80
incremental: N..../G84/U-1/W-41/D_3 1000/F80

Example 2
Longitudinal turning cycle with cut distribution

absolute: N.../G84/X13/Z-40/D_3 1000/F90
incremental: N.../G84/U-3.5/W-41/D_3 1000/F90

Cut distribution D_3 = 1mm (= 1000 µm)

3 x 1mm	3 mm
Rest (= finishing cut)	0,5 mm
Total advance	3,5 mm

Example 3
Longitudinal turning cycle with cut distribution and taper overmeasures

absolute: N.../G84/X14/Z-40/P_0 2.05/P_2 -2.66 /D_3 1000/F80
incremental: N.../G84/U-4/W-41/P0 2.05/P2 -2.66 /D3 1000/F80

G84 Face turning cycle

Input format:

N 4	G84	Z W ± 43	X U ± 43	P_0 P_2 ± 43	D_3 5	F 4
		[mm]	[mm]	[mm]	[µm]	[µm/r] [mm/min]

N ..... Block number
G84 .. Face turning cycle
Z(W) } Absolute (incremental) coordinates
X(U) } of the contour corner point (K)
P_0 ..... Taper dimension in X (radius value)
P_2 ..... Taper dimension in Z
D_3 ... Cut distribution
F ..... Feed

If there is no P_0 or P_2, it need not be programmed.

Difference between longitudinal turning and facing cycle:
With the longitudinal turning cycle X(U) is programmed before Z(W) - thus the first movement is carried out in X-direction.
With the facing cycle Z(W) is programmed before X(U) - thus the first movement is carried out in Z-direction. As for the rest, longitudinal turning and facing cycle are similar.

Longitudinal turning cycle G84 - X - Z	Facing cycle G84 - Z - X

Example 1
Facing cycle without cut distribution

absolute: N.../G84/Z-1/X6/D_3 1000/F80
incremental: N.../G84/W-1/U-8/D3 1000/F80

Example 2
Facing cycle with cut distribution

absolute: N.../G84/Z-2/X6/D_3 600/F100
incremental: N.../G84/W-2/U-8/D_3 600/F100

Cut distribution D_3 = 0,6mm (= 600µm)

3 x 0,6 mm	1,8 mm
Rest (= finishing cut)	0,2 mm
Total advance	2 mm

Example 3
Facing cycle with cut distribution and taper overmeasures

absolute: N.../G84/Z-3/X10/P_0-1.33/P_2-1.12 /D_3 700/F80
incremental: N.../G84/W-3/U-6/P_0-1.33/P_2-1.12 /D_3 700/F80

G85 Thread cycle

Input format:

N 4	G85	X U ± 43	Z W ± 43	P_2 ± 43	D_3 5	D_6 5	F 4
		[mm]	[mm]	[mm]	[µm]	[µm]	[µm]

N ..... Block number
G85 .. Thread cycle
X (U) } Absolute (incremental) coordinates
Z (W) } of the final thread point (K)
P_2 ..... Thread lead-out
D_3 ... Cutting depth of the first cut
D_6 ... Thread depth
F ..... Pitch

Definition
A cycle is a movement sequence of the tool which consists of a series of individual movements.

G00 Movement with rapid motion

Movement with programmed pitch (F) in µm/r

A cycle always has a closed movement sequence: starting point (S) = final point (E)
The starting point (S) is approached in the block before G85.

Contour corner point (K)

The movement sequence of a cycle is defined by starting point (S) and contour corner point (K). In accordance with the position of S and K to each other there are four kinds of cycle that you get as a result.
K is described with X,Z absolute or with U,W starting from S (incremental).

External right-hand thread

External left-hand thread

Internal right-hand thread

Internal left-hand thread

Thread lead-in (A) - thread lead-out (P_2)

During start of thread-cutting the Z-slide has to be accelerated. At the end of the thread the Z-slide has to be decelerated. The thread pitch is not constant in the acceleration and deceleration phase. Therefore this phase must be placed ouside the workpiece.

The starting point (S) has to be determined in such a way that there is a necessary thread lead-in (A) of approx. 3 to 4 mm.
For the thread lead-out (P_2) approx. 1 to 2 mm should be programmed. The threading tool moves out obliquely.

Thread depth (D_6)

You can find the thread depth for the turning tools used here in the tables of the chapter "thread-cutting".

Threading depth of the first cut (D_3)

A thread is always cut in several sequences. The first threading depth is programmed with D_3 in µm.

The further threading depths are calculated by the computer in such a way that the chip cross-section remains constant.
The advance for the individual cuts is carried out in X direction only.

Thread pitch (F)

The thread pitch is programmed in µm;
e.g. 1,25 mm pitch; input = F1250

You can find the right thread pitch for standardized threads in the tables of the chapter "thread-cutting".

Example:

absolute: N.../G85/X7,8/Z-20/P_2-2/D_3 600 /D_6 1100/F1500

incremental: N.../G85/U-1,6/W-23/P_2-2/D_3 600 /D_6 1100/F1500

G86 Cut-in cycle

Input format:

N 4	G86	X U ± 43	Z W ± 43	D_3 5	D_5 5	F 4
		[mm]	[mm]	[µm]	[µm]	[µm/r] [mm/min]

N ..... Block number
G86 .. Cut-in cycle
X(U) } Absolute (incremental) coordinates
Z(W) } of the contour corner point (K)
D_3 ... Advance per cut (radius value)
D_5 ... Tool width
F ..... Feed

The cut-in cycle consists of several single motions.

⇨ G00 Movement with rapid motion speed

→ G01 Movement with programmed feed speed (F) in mm/min or µm/r.

Contour corner point (K)
The cut-in depth and width is defined by starting point (S) and contour corner point (K). S is traversed in a block before G86.
K is described with X, Z absolute or with U,W from S (incremental).

Cut-in width = tool width
The tool movement is carried out in X direction. The tool advances by a measure of D_3, moves a little back (for chip breaking), advances again by a measure of D_3 etc., until point K is reached.

Cut-in width larger than tool width
If the width of the programmed cut-in is larger than the tool width, the control system grades the remaining cut-in width after the first cut-in into partial cut-ins with the same chip width. Overlapping of the individual partial cut-ins is at least 1/10 mm.

Each cut-in movement (1,4,7) is graded additionally by D_3 (see previous drawing).

Tools used:

Parting-off tool:	Cut-in tool:
Tool width = 3mm	Tool width = 1,2 mm max. cut-in depth = 1,5

When adjusting the tool (chapter 4,5) the left cutting edge has to be adjusted.

Example 1
Cut-in for Seeger circlip ring
Speed with aluminium = 1500 rpm

absolute: N.../G86/X14.3/Z-17/D_3 850/D_5 1200 /F22mm/min
incremental: N.../G86/U-0,85/Z 1,2/D_3 850/D_5 1200 /F22 mm/min

Example 2
Cut-in with 8 mm width in aluminium
Speed = 950 rpm

absolute: N.../G86/X12/Z-24/D_3 800/D_5 300/F150 mic/U
incremental: N.../G86/U-5/W8/D_3 800/D_5 300/F150 mic/U

Cut-in values for sealing rings according to DIN 471

d	d_1	m
9	8,6	1,2
10	9,6	1,2
12	11,5	1,2
14	13,4	1,2
15	14,3	1,2
17	16,2	1,2
18	17	1,3
20	19	1,3
25	23,9	1,3
30	28,6	1,6
35	33	1,6
40	37,5	1,85
50	47	2,15
60	57	2,15

The reference point (R)

Position:
After setting up a blank, the reference point is automatically on the extreme right end of the blank on the center line.

Meaning:
All absolute dimension values on the workpiece refer to the reference point. By special dimensioning of the workpiece it may be useful to set off the reference point during machining.

Offset:
With G92 the offset values for the reference point (R) are indicated. With G59 the offset indicated in block G92 is activated.
With G56 the offset is reversed - R is again in the original position. G56 is not necessary at program end - when calling up a new program, the reference point offset is cancelled automatically.

G92 Data for reference point offset

Input format:

N 4	G92	X ± 43 U	Z ± 43 W
		[mm]	[mm]

X, Z (or U,W) are the offset values. The new reference point (R_1) is offset by this values from the old reference point (R).

If there in no reference point (R) the offset values refer to the machine starting point (M).

X is a diameter value
U is a radius value

Example:

absolute: N.../G92/X0/Z-30
incremental: N.../G92/U0/W-30

G94 / G95 Feed unit

Input format:

N4	G94

N4	G95

- Switch-on state is G94
- G94 and G95 are self-holding until they are cancelled
- G94, G95 can be programmed as often and wherever one wants to

For the feed size or feed speed the letter F is used.

1. Feed programming with G94
= path of turning tool per minute
F [mm/min]

2. Feed programming with G95
= path of turning tool per main spindle revolution
F [µm/r] ....1 µm = 1/1000 mm

Conversion:

F (mm/min) = S (rpm) x F (mm/r)

$$F\,(mm/r) = \frac{F\,(mm/min)}{S\,(rpm)}$$

S.... Main spindle speed

PROGRAM SHEET COMPACT 5 PC

N	G,M	X, U	Z, W	I	K	F	T	P0	P2	D3	D5, D6	REMARK

7. NOTES FOR THE SOFTWARE COMPILATION

Design:

The power supply unit and the stepping motor control of the Compact 5 PC are accommodated in a smaller control cabinet. These components are identical to those of the Compact 5 PC and therefore the same spare parts can be employed.
The Compact 5 PC must be controlled by an external computer. The Compact 5 PC is to be connected to this computer like a peripheral, such as a printer, plotter or a paper tape reader / puncher. The computer must be fitted with a Centronics interface for connection to the Compact 5-PC. This interface is a parallel interface, i.e. data is transmitted on eight lines simultaneously (Centronics is the company name of an American printer manufacturer and the definition of this printer interface has become a quasi-standard worldwide).
The connector provided on the Compact 5 PC is a 25-pin D subminiature jack connector with the following pin assignment:

Pin No.	Signal designation
1	STROBE
2	data bit 0
3	data bit 1
4	data bit 2
5	data bit 3
6	data bit 4
7	data bit 5
8	data bit 6
9	data bit 7
10	speed pulse light barrier (100 p/rev.)
11	emergency-off and chip protection door
12	synchronous pulse light barrier
18 - 25	earth

This interface is connected via a small interface pc board directly to the stepping motor card and the light barrier. This interface board accommodates a 4-fold DIP switch with which the user can set various modes of operation.

Signal description:

The signal inputs only process TTL signals and the signal outputs offer TTL signals.
The strobe input serves for the synchronisation of the data lines. If a pulse is transmitted on this line, the data which are offered at this time to the data lines bit 0 - bit 7 are accepted as being valid. On many computers this signal is automatically produced by the hardware, on other computers the programmer has to generate this signal in the software.
The signal emergency-off and chip protection door is a signal output and is set to "Low" when the emergency-off button is pressed or if the chip protection door is open. (The limit switch for the chip guard is <u>not</u> installed as a standard feature. Merely the connection for it is provided!).

The signal speed pulse is a signal output and provides 100 pulses per revolution of the main spindle. The pulse duty factor is not defined and also depends on the speed.
The signal synchronous pulse is a signal output and provides one pulse per revolution of the main spindle. The length of the pulse depends on the speed and the pulse is not synchronised with the slopes of the speed pulses.
The meaning of the data signals depends on the setting of the 4-fold DIP switch. These signals, however, always serve to trigger the stepping motors.
In order to picture the operation of the stepping motor, it is a good idea to imagine a coordinate system with the four cardinal points. These points correspond to the directions of the magnetic fields in the stepping motor and one magnetic field can be produced with one winding (when an electric current is allowed to pass through the winding). If more than one winding is switched on at the same time, the direction and strength of the magnetic field are derived by vectorial addition. An example: If the windings "North" and "South" are activated at the same time, the two magnetic fields cancel each other out and the resultant magnetic field has zero strength and the direction is undefined. However, if the windings "South" and "West" are switched on simultaneously, the direction of the resultant magnetic field is "South-West" and the strength is 1.41 times that of a single field. (Diagonal of a square).

The rotor of a stepping motor is designed so that it always tries to align itself in the direction of the magnetic field produced by the stator. Therefore, if the current is suitably switched on and off in the windings of the motor, the rotor rotates about a certain angle and it thus moves stepwise from one set magnetic field direction to the next.
Ther are two ways in which the windings can be triggered:

1) Full-step mode	1st step	S-W
	2ndstep	W-N
	3rd step	N-E
	4th step	E-S
2) Half-step mode	1st step	S-W
	2ndstep	W
	3rd step	W-N
	4th step	N
	5th step	N-E
	6th step	E
	7th step	E-S
	8th step	S

The direction of rotation depends on whether one controls from 1 to 2 etc., or, for example, from 1 to 8, from 8 to 7 etc.
The advantages of the half-step mode of operation lie in the double resolution with the same motor design, the disadvantage is a loss of torque because only one winding is active at each intermediate step and only about 70% of the maximum torque is available. Moreover, more stringent demands have to be placed on the computing speed owing to the higher resolution.

The Centronics interface has eight data lines. As four windings per motor have to be triggered, the windings can be directly controlled by two motors.
If the sequence of winding control no longer has to be performed directly by the connected computer, two lines per axis drive are, in principle, sufficient, i.e. one line where the direction of rotation is determined and a second line where the steps are output. (One pulse on the second line means that the stepping motor is to move one step in the direction or rotation specified on line one.)

Abbreviations:

VS ...	full step
HS ...	half step
T/R ..	step/direction signal
dir ..	direct winding control
RL ...	cw
LL ...	ccw

Table for the 4-fold DIP switch:

Mode				1	2	3	4
VS	T/R	RL	RL	on	on	on	on
VS	T/R	LL	RL	on	off	on	on
VS	T/R	RL	LL	on	on	off	on
VS	T/R	LL	LL	on	off	off	on
VS	dir	RL	RL	off	on	on	on
VS	dir	LL	RL	off	off	on	on
VS	dir	RL	LL	off	on	off	on
VS	dir	LL	LL	off	off	off	on
HS	dir	RL	RL	off	on	on	off
HS	dir	LL	RL	off	off	on	off
HS	dir	RL	LL	off	on	off	off
HS	dir	LL	LL	off	off	off	off
HS	T/R	RL	RL	on	on	on	off
HS	T/R	LL	RL	on	off	on	off
HS	T/R	RL	LL	on	on	off	off
HS	T/R	LL	LL	on	off	off	off

Characterisitics of the axis drives for the Compact 5 PC:
The specified values relate to full-step control and are to be converted for half-step control accordingly.
The given values relate to the slide movements.

Resolution:
One step corresponds to 1000/72 micrometre.

Speed:
100 Hz correspond to 83.33 mm/min feed rate

Maximum speed:
840 Hz corresp. to 770 mm/min rapid motion

Start-stop frequency:
598 Hz corresp. to 499 mm/min feed rate

Acceleration time:
from 0 to 598 Hz 0 sec
from 598 to 840 Hz 1 sec (linear acceleration)

"dir" mode:
The data lines D0 to D3 are assigned to the stepping motor in the X-axis (cross slide). The sequence of the windings is "E", "S", "W", "N".
The data lines D4 to D7 are assigned to the stepping motor in the Z-axis (longitudinal slide). The sequence of the windings is the same as for the X-axis.

"T/R" mode:
The data lines

D0 is direction motor X
D1 is steps motor X
D2 is direction motor Z
D3 is steps motor Z

Menu summary

Notes

You can select and activate the individual menu points with the function keys F1 to F10 or with the space bar.

A command can be interrupted or a return to the previous menu level activated with the key Esc.

Hotkeys - draw

In order to facilitate a more efficient programming, so-called "hotkeys" were introduced in this software. Hotkey = single-key commands, which are effective in every menu level. You can display a work piece contour on the screen more quickly with the use of these hotkeys.

Example: Constructing a line

Possibility 1 - Menu
1. F2 Recall - menu DRAW
2. F2 Recall - submenu POINT
3. Moving the cursor to the starting point
4. F1 Determining the starting point
5. Moving the cursor to the end point
6. F2 Determining the end point
7. Esc. Exit from the submenu POINT
8. F3 Recall - submenu line
9. F1 Drawing a line

Possibility 2 - hotkeys
1. Moving the cursor to the starting point
2. S Determining the starting point
3. Moving the cursor to the end point
4. + Drawing the line to the cursor position, a new starting point at the same time

Summary - hotkeys

Key	Cursor commands
I	Position the cursor incrementally
/	Position the cursor by entering the radius and angle
Pos 1	Cursor to the middle of the screen
End	Cursor to the reference point
PgUp↑	Doubling the step size of the cursor
PgDnf	Halving the step size of the cursor

Key	Screen commands
N	Redrawing the screen contents
W	Zooming a screen window
Z	Zooming the work piece
>	Cursor as a graticule
<	Reduced cursor

Key	Searching commands
P	Finding the point
L	Searching for the line
A	Searching the circle, arc
X	Searching any point of intersection

Key	Point commands
S	Determining the starting point
E	Determining the end point
M	Determining the centre point

Key	Geometric commands
R	Adding a fillet
C	Adding a chamfer (45°)
Y	Cleaning the bit
+	Drawing a line to the cursor
-	Erasing the element

Note:
Letters can be entered as capitals or small letters.

Hotkeys - machine

You can move the slides and influence the machining cycles by means of these keys.

↑	Move the slides in the -X direction
↓	Move the slides in the + X direction
←	Move the slides in the -Z direction
→	Move the slides in the + Z direction
PgUp↑	Enlarge the travel path of the stepping motors
PgDn↓	Reduce the travel path of the stepping motors
>	Increase the feed
<	Reduce the feed
Space bar	Stop the slides (= intermediate stop)
Esc.	Interrupt the machining

Travel path of the stepping motors

PgUp↑	10 mm (= largest travel path)
PgUp↑	5 mm
	1 mm (= switch-on condition)
PgDn↓	0,5 mm
PgDn↓	0,1 mm
PgDn↓	0,05 mm
PgDn↓	0,01 mm (= smallest travel path)

Altering the programmed feed

>	200 %
>	150 %
>	120 %
	100 % (= switch-on condition)
<	80 %
<	60 %
<	50 %
<	40 %
<	30 %
<	20 %
<	10 %
<	5 %
<	0 %