emco group

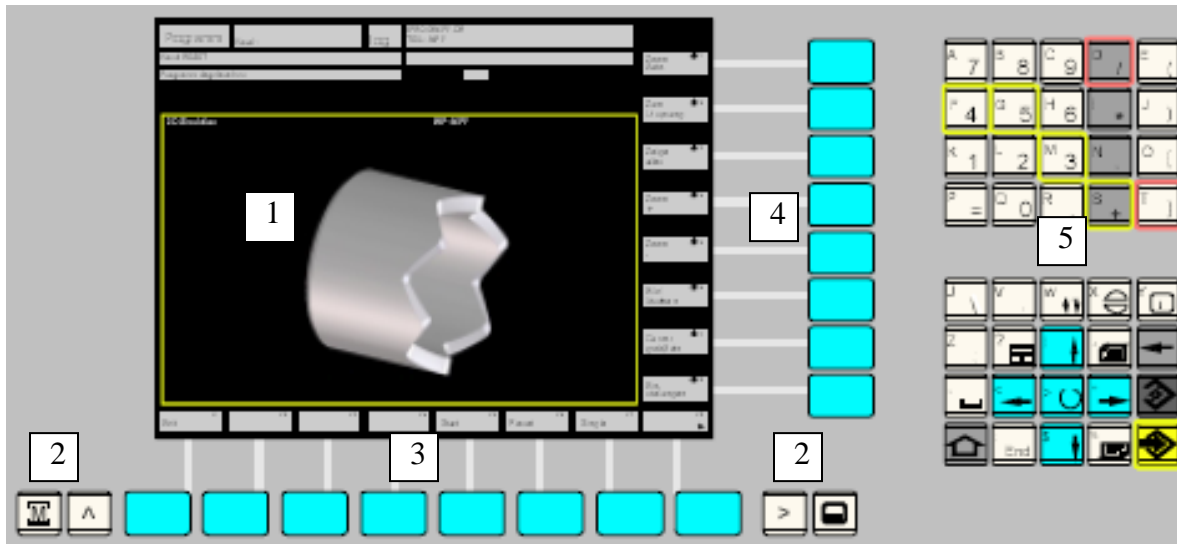**Siemens
Sinumerik 810D / 840D
Training Guide**

SIEMENS 810D/840D CONTROLS

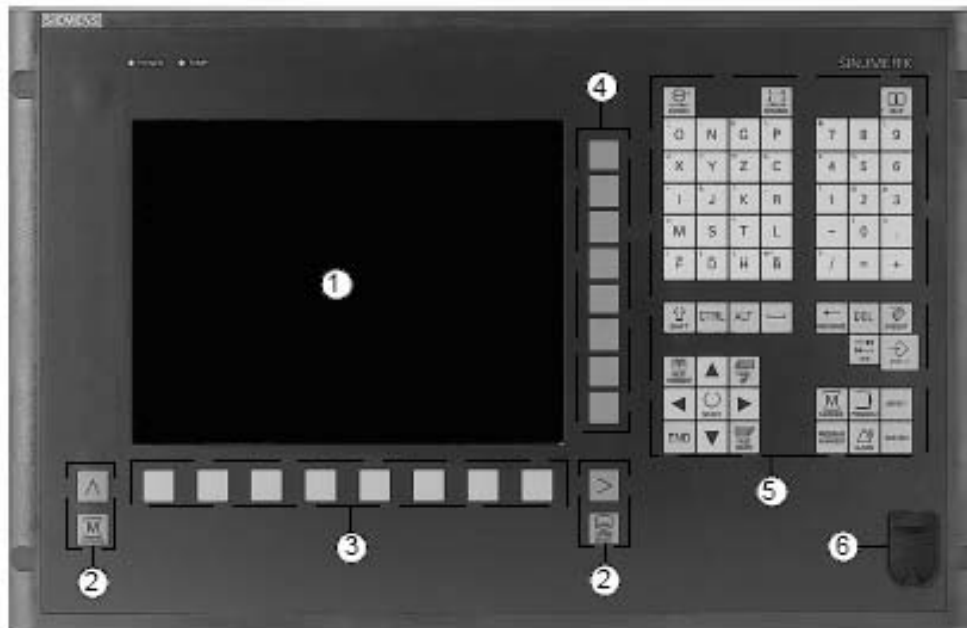## EMCO PREVIOUS CONTROL SERIES

Operator panel OP 031

1. **Monitor**
2. **Monitor Keys**
3. **Horizontal Soft Keys**
4. **Vertical Soft Keys**
5. **Alphanumeric Pad**
   Correction/cursor pad with control keys and input keys
6. **Next Page USB connection**

CONTINUED SIEMENS 810D/840D CONTROLS

## EMCO ET CONTROL SERIES

Operator panel OP 010

EMCO HT / MT CONTROL SERIES

Operator panel OP 010C

SIEMENS 810D/840D SCREEN

Machine **1**	Channel 1	**2**	AUTO**3**	**4**		Trans./G function
Channel reset		**5**		**6**		
Program aborted	**7**		ROV **8**			

9

WCS	Position		Dist-to-go		Master spindle	S1	Auxiliary function

X	3.2283	inch	0.0000	Act.	0.000 rpm		Spindles
Z	3.1890	inch	0.0000	Set	0.000 rpm		
				Pos	0.0000 deg		
	10			**10**	100.0000 %		Axis feedrate
				Power [%]			

11

Actual block		Feedrate	inch/min	Program blocks **12**
		Act.	0.000 100.0000 %	
		Set **10**	0.000	Zoom act. val.
10		Tool		
		Preselected tool: **10**		Act. val. Mach(MCS)
		T-1		
14		G00 G40		Program level

13   **15**

16	Program control	Block search		Correct program	Program overview **17**

1. Display of the active Operating Area
2. Display of the active channel
3. Operating mode, when a sub mode is active, it also will be displayed (e.g. REF, INC)
4. Program path and name of the selected program
5. Channel status
6. Channel operating messages
7. Program status
8. Channel status display (SKIP, DRY, SBL)
9. Alarm and message line
10. Working window, NC display. The working windows (program editor) and NC displays (feedrate, tool) available in the active Operating Area are displayed here.
11. The selected window is marked with a border and the headline is displayed inverted. The keyboard inputs are effective here.
12. Vertical soft keys These 8 fields show the functions of the keys right beside. (at the PC: Shift F1..F8)
13. Dialog line with operator prompts
14. Recall function; When this symbol is displayed, the key ⋀ is active (jump back to superior menu).
15. When this symbol ⅰ is displayed, the key is active (information available).
16. Horizontal soft keys These 8 fields show the functions of the keys below. (at the PC: F1..F8)
17. ETC. Function; When this symbol is displayed, the key ▣ is active (more soft key functions available in this line).
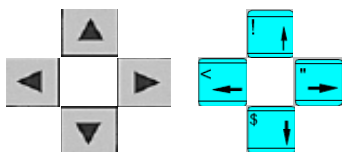
SIEMENS 810D/840D KEYS

New   Old   = keys for the Control Series of Siemens

= Channel keys; switch Turret systems

= Confirm alarm or cancel alarm keys

= Show information for the actual operating status - works only when the dialogue line shows an [i] .

= Window selection key; switches to other viewable windows
Keyboard inputs are valid for the selected window only.

= Cursor down / up

= Cursor left / right   **OLD**

= Leaf backward or page down

= Leaf forward or page up

NEW

= Blank

= Clear (Backspace)

= Selection key / Toggle key

- Selection of predefined input values in input fields and lists, which are marked with this symbol
- Activate / deactivate switch box / radio button

⊠ ⦿   = active
☐ ○   = not active

= Edit key / Undo
- Switch to edit mode in tables and input fields
- Undo function for table elements and input fields (leaving a filed with this key does not store the entered value but reestablishes the old value)

= End Jump to line end (list end)

New   Old   = keys for the Control Series of Siemens

= **Input key**
- **Take over an edited value**
- **Open / close directory**
- **Open file**

= **Shift key**   **OLD**

= **Machine Key**

= **Page Back**   **NEW**

= **Page Over**

= **Main Pages or Last Page**
- **Machine, Parameter, Program, Services, and etc.**

New   = keys for the Control Series of Siemens

= **Same as Windows but works only on a PCU 50**

= **Deletes characters, numbers**

= **Used to jump between areas or selections**

= **Machine Key**   **NEW**

= **Program Key**

= **Offset Key**

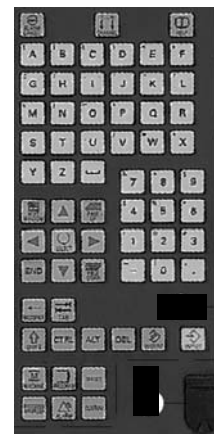= **N/A (Some Controls use this for Shop Turn)**

= **Editor for active main program**

= **Alarm Key**

5

ET420   EMCO Machine Control

Operator Keys   Directional Keys   Spindle Keys   Accessory Keys   MODE Dial   Feed Override Dial

E - Stop   Special Operation   Protection Key   Cycle Start   Tool System Key   Agreement Button

HT665   EMCO Machine Control

Operator Keys   Directional Keys   Spindle Keys   Accessory Keys   MODE Dial   Feed Override Dial

E - Stop   Special Operation   Protection Key   Cycle Start   Main Spindle Chuck Button   Sub Spindle Chuck Button   Agreement Button

MT, ET 25   EMCO Machine Control

Directional Keys | Feed Override Dial | Operator Keys | MODE Buttons | Accessory Keys | Spindle Keys

E - Stop | Cycle Start | One Times Button Single Piece | Main Spindle Chuck Button | Special Operation | Protection Key | Agreement Button

MT, ET 45 / 65   EMCO Machine Control

Operator Keys | MODE Buttons | Directional Keys | Accessory Keys | Spindle Keys | Feed Override Dial

E - Stop | Special Operation | Protection Key | Cycle Start | One Times Button Single Piece | Main Spindle Chuck Button | Sub Spindle Chuck Button | Agreement Button
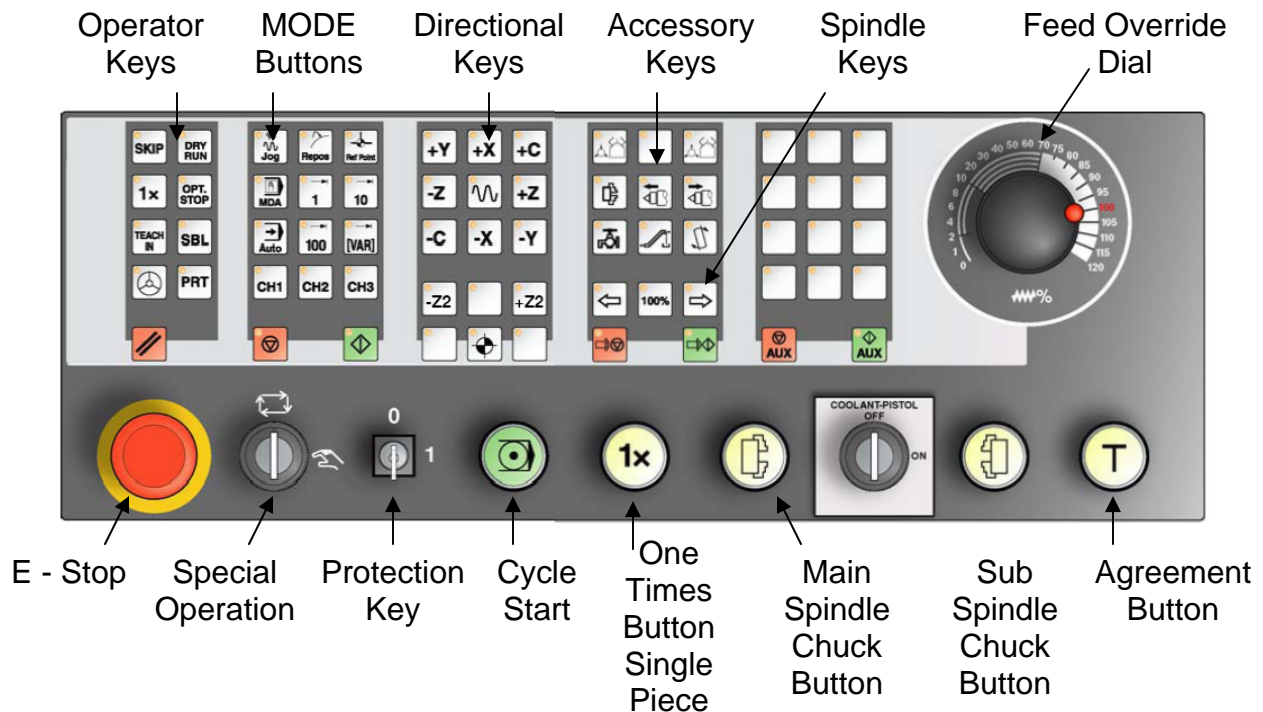
EMCO MACHINE KEYS

SKIP = Press skip for any block lines with (/) (Slash) before block number will be skipped

DRY RUN = Press for test run without spindle on and rapids only (remove raw material from vise)

1 x = (Single piece) for continuous mode active only on automatic material loading

OPT. STOP = (Optional stop) for programs with (m1)

// = (Reset) cancels most alarms, resets program, interrupts programs

S BL = (Single block) reads one block line at a time

PRT = (Program Test) allows simulation without machine moving
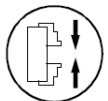
⊘ = (Cycle stop) program hold, feed hold

◇ = (Cycle start) program start

Large Buttons and Keys

= **Additional Chuck Button**
Hand Mode is for moving machine around with door open and works in conjunction with the (Agreement button)

= **(Agreement button)** used for open/closing door or to jog axis with the door open
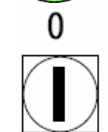
= **(Mode Key) Automatic & SETUP (Hand Mode)**
Setup (Hand Mode) is for moving machine around with door open and works in conjunction with the (Agreement button)
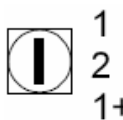
= **(Cycle start) program start**

= **Data Protection**
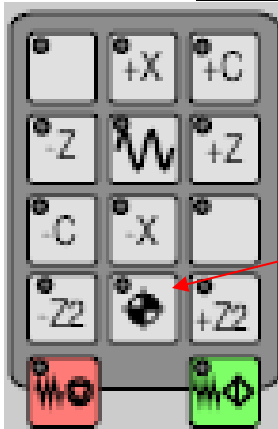Position 0 = Locked   Position 1 = Unlocked

= **Tool System Key**
1 = Upper System   2 = Lower System   1 + 2 = Both

DIRECTION KEYS

These keys control axes directional movements

+C & -C = Rotary axes

Reference all

Z2 = Sub Spindles

Feed stop (Red) / Feed start (Green)

SPINDLE OVERRIDE KEYS

Arrow key pointing right increase the Spindle speed (120% high)

Arrow key pointing left decrease the Spindle speed (50% low)

100% key jumps speed to 100%

Spindle stop (Red) / Spindle start (Green)

ACCESSORY FUNCTIONS
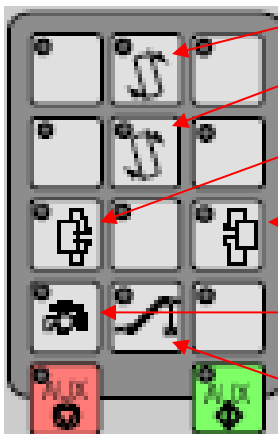
Press once upper turret index
Press once lower turret index

Main Collets press once chuck open
Main Collets press again chuck closed

Sub Collets press once chuck open
Sub Collets press again chuck closed

Press once coolant on
Press again coolant off

Press once Chip Conveyor on
Press again Chip Conveyor off

Press auxiliary drives on (Green)
Press auxiliary drives off (Red)

FEED OVERRIDE DIAL
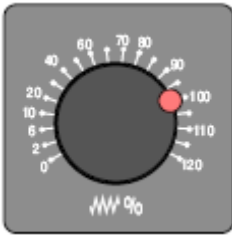
Controls feed for jogging in the X, Z and C Axis.
Overrides from 0% to 120% of the programmed
feed rate or the rapid rate

Coolant Pistol Control
On and Off switch
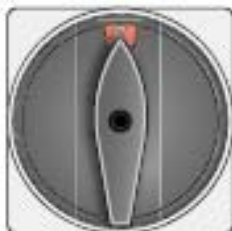
Main Switch
0 = Off
1 = On
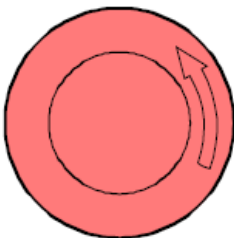
E Stop or Emergency Stop

Interrupts all drive including spindle. Needs re-referenced to continue

MODE DIAL
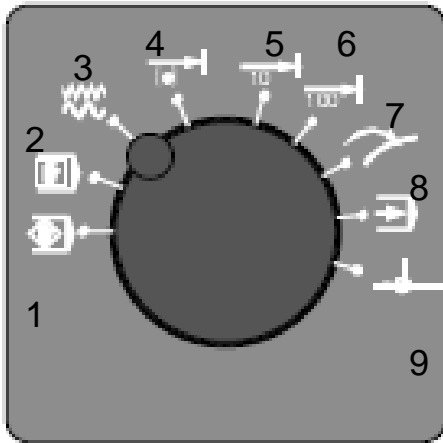
(1) Teach In = Creating programs in Buffer (MDA)

(2) MDA = Manual Data Automatic mode for manually running the machine

(3) JOG = Manual moving the axis in X, Z and C

(4) INC Var = Setting value for incremental steps

(5) INC = incremental steps 10 (.0010/ten thousands)

(6) INC = incremental steps 100 (.0100/hundred thousands)

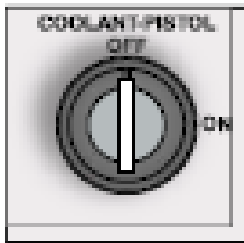**(7) Repos = approach back to a contour**

(8) Automatic = mode for running a program

(9) Ref = Referencing the machine or homing the machine

INC = incremental steps 1 (.0010 / thousands)

Hand Wheel = Allows hand wheel to move axis

Tail Stock = Tail stock (Quill) Forward and Back

Power Up / Down and Status

Power Up
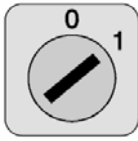
1. **Side of the Machine rotate switch to 1 = On**

2. **Open then Close door using ⓉT button**

3. **Front of the machine Pull / Twist E-Stop out**
   - **Some Machines need Reset Pressed**

4. **Press and hold for 3 to 5 seconds Aux = On**

Power Down

1. **Press Aux = Off**

2. **Front of the machine Push E-Stop in**

3. **Side of the Machine rotate switch to 0 = off**

Status Lamp
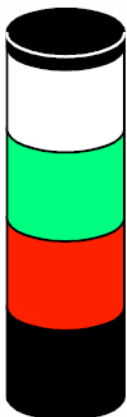
White = Lack of parts

Green = Automatic active

Red = Alarm or E Stop

Starting a Work folder

1. Press the ⬛ button on the horizontal soft keys **(press** ⬛ **again if picture 1 soft key don't appear)** now press Blue horizontal soft key for [Program F3]

2. If Work-pieces page is not highlighted press Blue horizontal soft keys for [Work-pieces F1]

3. Press the Blue vertical soft key for [New ⇧F1]

4. Type in 123456 then press the Blue vertical soft key for [OK ⇧F8] **(This is the folder name that holds the main and sub programs)**

5. Press the input ◈ button (this is to open / edit the folder to place programs into)

Picture 1

Picture 2
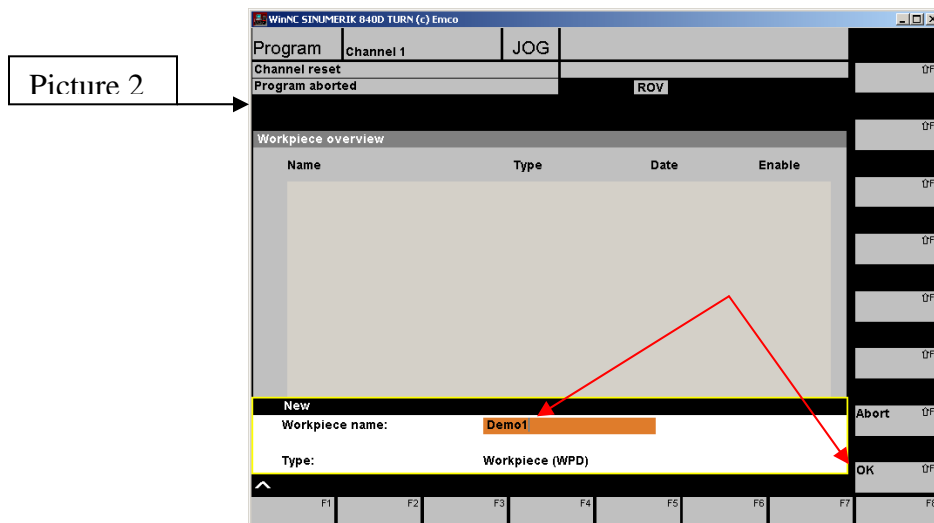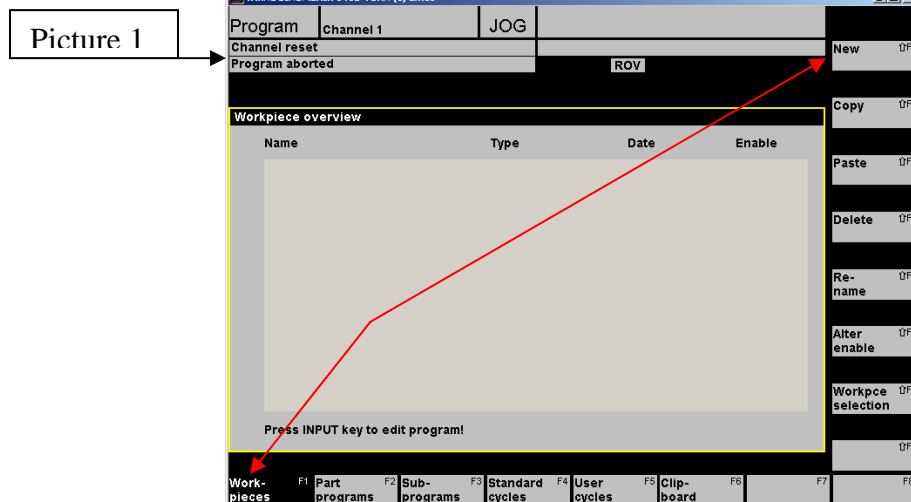
Starting a Main Program File (MPF)

1. Press the Blue vertical soft key for **New** ⇧F1

2. Highlight Program name and type Main

3. Cursor down to Type and press [icon] until part program (MPF) appears (this is the name created for the main program file)

4. Press the Blue vertical soft key for **OK** ⇧F8

5. Press the Blue vertical soft key for **Close** ⇧F8

6. Press the Blue vertical soft key for **Program selection** ⇧F7 (this make the demo 1 as the main program that is active)

Starting a Sub Program File

1. Press the Blue vertical soft key for **New** ⇧F1

2. Highlight Program name and type OD 1

3. Cursor down to Type and press [icon] until subprogram (SPF) appears (this is the name created for the sub program file)

4. Press the Blue vertical soft key for **OK** ⇧F8
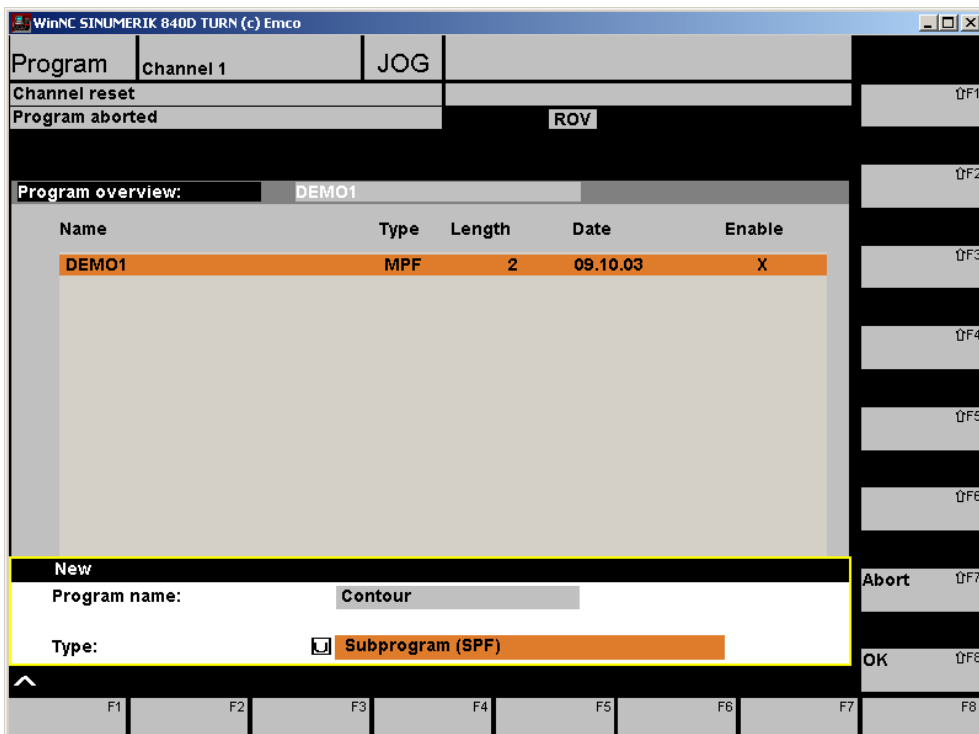
M00	programmed stop
M01	optional stop
M02	end of program
M03	main spindle ON clockwise (SXXXX...speed)
M04	main spindle ON counterclockwise (SXXXX...speed)
M05	main spindle OFF
M2=03	counter spindle ON clockwise (SXXXX...speed)
M2=04	counter spindle ON counterclockwise (SXXXX...speed)
M2=05	counter spindle OFF
M3=03	driven tool ON clockwise (S3=XXXX ...speed)
M3=04	driven tool ON counterclockwise (S3=XXXX ...speed)
M3=05	driven tool OFF
M07	high pressure coolant pump ON
M08	coolant pump ON
M09	coolant pump OFF
M10	main spindle brake ON
M11	main spindle brake OFF
M2=10	counter spindle brake ON
M2=11	counter spindle brake OFF
M17	end of subprogram (RET)
M20	tailstock backward
M21	tailstock forward
M22	eject finished part
M23	parts catcher backward
M24	parts catcher forward
M25	open main spindle chuck
M26	close main spindle chuck
M2=25	open counter spindle chuck
M2=26	close counter spindle chuck
M30	end of main program
M30 M32	restart machine to be programmed with M30
M42	Start channel 2
M50	bi-directional tool-turret function off
M57	main spindle oscillating ON
M58	main spindle oscillating OFF

M59	command to set dwell for chip conveyor (H...time in sec)
M60	open gantry door
M61	close gantry door
M31	(release gantry door)
M65	Feed hold/stop read in for KUPA/EMCO bar loader
M66	cancel feed hold/stop read in for KUPA/EMCO bar loader
M67	feed command ON for bar feeder (only in automatic made)
M68	feed command OFF for bar feeder (only in automatic made))
M69	change bar
M71	start to turn spindle 50rpm for cleaning (M8/M7 has to be programmed)
M72	M71 OFF
M73	stopwatch ON (R99)
M74	stopwatch OFF (R99)
M81	channel 1 wait function for M30 in channel 2
M89	spindle rotation with open chuck
M90	Reset M89
M96	set „REQUIRED NO. OF PIECES"
M99	TO SET MACHINE CONFIGURATION (with H-command)
M150	programmable output signal (A57.0) ON
M160	programmable output signal (A57.0) OFF
M151	programmable output signal (A57.1) ON
M161	programmable output signal (A57.1) OFF
M152	programmable output signal (A57.2) ON
M162	programmable output signal (A57.2) OFF
M153	programmable output signal (A57.3) ON
M163	programmable output signal (A57.3) OFF
M154	programmable output signal (A57.4) ON
M164	programmable output signal (A57.4) OFF
M155	programmable output signal (A57.5) ON
M165	programmable output signal (A57.5) OFF
M156	programmable output signal (A57.6) ON
M166	programmable output signal (A57.6) OFF
M157	programmable output signal (A57.7) ON
M167	programmable output signal (A57.7) OFF

Overview of M-functions in channel 2

M17	end of subprogram (RET)
M25	open main spindle chuck
M26	close main spindle chuck
M2=25	open counter spindle chuck
M2=26	close counter spindle chuck
M30	end of main program
M30 M32	restart machine to be programmed with M30
M41	Start channel 1
M60	open gantry door
M61	close gantry door
M31	(release gantry door)
M65	open raw-part gripper
M66	close raw-part gripper
M67	open finished part gripper
M68	close finished part gripper
M71	Wash on (Spindle 50rpm M8/M7 has to be programmed)
M72	Wash OFF
M83	start pallet magazine forward
M84	start pallet magazine backward
M85	start pallet magazine for special fixtures
M89	spindle rotation with open chuck
M90	Reset M89
H1	raw part gripper left - finished part gripper right
H2	raw part gripper down - finished part gripper up
H3	raw part gripper right - finished part gripper left
H4	raw part gripper up - finished part gripper down

Channel 1(machine):

H0 M99	manual chuck on main spindle
H1 M99	OD clamping at the main spindle
H2 M99	ID clamping at the main spindle
H4 M99	bar feed or magazine ON
H5 M99	bar feed or magazine OFF
H10 M99	manual chuck on counter spindle
H11 M99	OD clamping on counter spindle
H12 M99	ID clamping on counter spindle

channel 2 (gantry):

H1 M99	raw part gripper OD clamping
H2 M99	raw part gripper ID clamping
H11 M99	finished part gripper OD clamping
H12 M99	finished part gripper ID clamping

INIT(1,"NAME") Read-in command (for channel 2) for the machining program allocated to channel 1

INIT(2,"NAME") Read-in command (for channel 1) for the machining program allocated to channel 2

TRANS Translation: absolutely programmable offset

ATRANS Additive translation: incrementally programmable offset

ROT Rotation: absolutely programmable rotation

AROT Additive rotation: incrementally programmable rotation

SCALE: absolutely programmable scaling

ASCALE Additive scale: incrementally programmable scaling

MIRROR: absolutely programmable mirroring

AMIRROR Additive mirror: incrementally programmable mirroring

Note: absolute offsets cancel all other selected frames

SETMS(n) Determination of a spindle as master spindle (to which all programmed commands refer)

SPOS[n]=0 Positioning of a spindle on 0° (in every program at the first selection of a spindle as rotary axis (C- axis) compulsory for its referencing!)

STOPRE Read-in stop: prevents further advanced reading of NC blocks until the last function before this command has been carried out.

TMCON Face milling work at the main spindle ON (TRANSMIT ON)

TMC2ON Face milling work at the counter spindle ON (TRANSMIT ON)

TRCON(Dm) Cylinder interpolation at the main spindle ON (TRACYL ON)

TRC2ON(Dm) Cylinder interpolation at the counter spindle ON (TRACYL ON)

dm = cylinder diameter before developing the surface area) Survey frames and specific commands

TMCOFF Cylinder interpolation OFF (TRACYL OFF) Face milling OFF (TRANSMIT OFF)

COUPDEF(.,.,.,.,.) Definition of the spindle synchronization with indication of the sequential spindle, leading spindle, sense of rotation (+/-) with speed relation:

COUPDEF(S3,S1,-2,1,"IPO","DV"); (e.g. polygonal turning)

SPCON(n) Determination of the leading and sequential spindle (in two subsequent NC blocks

SPCON(1); SPCON(3); (e.g. polygonal turning)

COUPON(Sn,Sn,w) Switch-on of the synchronization with indication of the sequential spindle, leading spindle, angle offset to each other COUPON(S3,S1,90); (e.g. polygonal turning)

COUPOF(Sn,Sn) Switch-off of the synchronization with indication of the sequential spindle, leading spindle

COUPOF(S3,S1); (e.g. polygonal turning)

SPCOF(n) Deselection of the leading and sequential spindle (in two subsequent NC blocks)

SPCOF(1); SPCOF(3); (e.g. polygonal turning)

DIAMON Diameter programming ON

DIAMOF Diameter programming OFF

FFWON Precontrol ON

FFWOF Precontrol OFF

BRISK Discontinuous path acceleration

SOFT Back-limited path acceleration

SUPA (as G53) Deselection zero point offset on machine coordinate system incl. frames [effective block wise]

MSG Programmable message e.g. {MSG ("END TOOL SERVICELIFE2");}

START(1) Start channel 1 (from channel 3 or channel 2)

START(2) Start channel 2 (from channel 3 or channel 1)

WAITE(1) Channel 3 or channel 2 waits until M30 in channel 1

WAITE(2) Channel 3 or channel 1 waits until M30 in channel 2

WAITM(m,k,k) Waits until label (= same command) reached in the other channel
{WAITM(1,2,1); (channel1 waits for channel2

until label 1 reached)}

SETM(m) Sets the marker "m" in the same channel without affecting current processing.
SETM() remains valid after RESET and NC-Start. It also can be programmed independently of a synchronized action.

CLEARM(m) Deletes the markers "m" in the same channel without affecting current processing. All markers can be deleted with CLEARM().

CLEARM() remains valid after RESET and NC-Start. It also can be programmed independently of a synchronized action.

LIMS Speed limitation for const. cutting speed {LIMS=1500;}

CFC Constant feed at the contour

CFTCP Constant feed at tool cutter reference point

CFIN Constant feed with internal curve

RPL Rotation of the selected working plane {AROT RPL=90;}

REPEAT mk1 mk2 Repetition of program parts between jump address 1 and
2)

GOTOF ABC Jump instruction forward till address ABC: (at least 3 characters)

GOTOB UVW Jump instruction backward till address UVW: (at least 3 characters)

ABC: Jump instruction (Attention: is considered without colon {ABC} as subprogram name!)

MCALL Modal cycle and/or subprogram call-up

MCALL BOHR; Deselection: command call-up without subprogram MCALL;

TURN Number of complete circles for thread milling (2,5-Dinterpolation)

G2 X-10 Y0 Z-10 I=AC(0)

J=AC(0) TURN=2;

RET (as M17) subprogram END

AC Programming with absolute indication of coordinates {X=AC(0);}

IC Programming with incremental indication of coordinates

X=IC(5); Z=AC(-5);

DC Approach coordinate directly {C=DC(300);}

ACP Approach position with absolute indication of coordinates in positive direction
C=ACP(300);

ACN Approach position with absolute coordinate indication in negative direction
C=ACN(300);

ANG Angle programming starting from zero beam (right, 3 o' clock) ausgehend,
counterclockwise (positive)
ANG=90;}

AP Polar angle {AP=120;}

RP Polar radius {RP=30;}

AR Opening angle {AR=100;}

CR Circular polar angle {G2 X... Z... CR=20;}

RND Insert transition radius {RND=2;}

RNDM Modal call-up for transition radius at all edges {RNDM=1 ; deselecting:
RNDM=0;}

CHR Insert transition chamfer as cathetus {CHR=2;}

CHF Insert transition chamfer as Hypotenuse {CHF=4;}

L700 Subprogram call-up for work piece counter (EMCO user program)

BAREND End of bar (parameter R0 is set "1")

BARFEEDDWELL Waiting, until loader Magazine has feed

BARCHANGEDWELL Waiting of start after bar changing

Cycles of the "Turning" group

Turning/drilling/tapping (G18):
tool type 500, L1=X, L2=Z
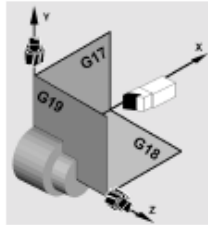
For standard turning and drilling, tool type "500" should be used in the standard coordinate system, since here only 2 lengths are queried.
The working plane remains at G18 and it is not necessary to change it for radial and axial machining.

Stock removal Cycle "CYCLE95"

This cycle can be used for external or internal, longitudinal or transversal roughing or finishing operations. The largest diameter for external roughing and the smallest diameter for internal roughing sets the start point for the roughing cycle.

The last position in your program must be out of the roughing area made by the cycle, otherwise an alarm "wrong startpoint" appears.
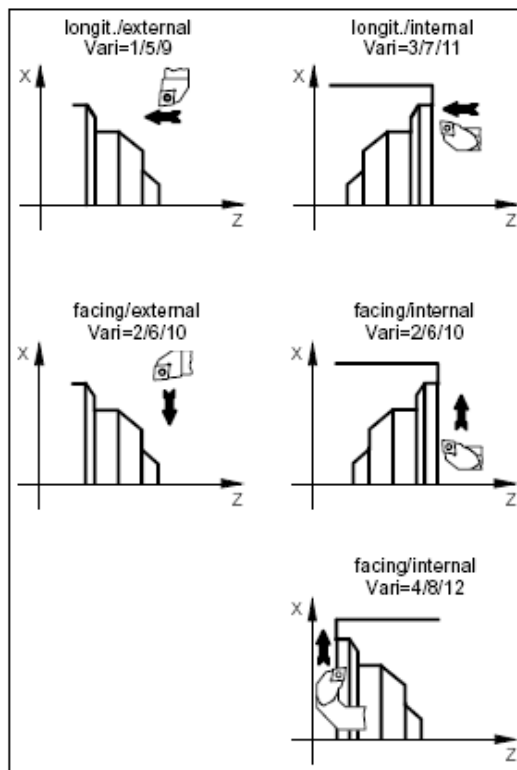
Types of machining

Type of machining

value	longitud. / facing	outside / inside	kind of machining
1	l	o	roughing
2	f	o	roughing
3	l	i	roughing
4	f	i	roughing [1]
5	l	o	finishing
6	f	o	finishing
7	l	i	finishing
8	f	i	finishing [1]
9	l	o	compl. machining
10	f	o	compl. machining
11	l	i	compl. machining
12	f	i	compl. machining [1]

1) Only used for the machining behind a shoulder

Example - longitudinal turning external contour

Name of the contour program .... "CONTOUR1"
Selection[1] Machining (complete, roughing or finishing) .... complete
Selection[1] (machining direction long. or face) .... long.
Selection[1] (type of machining outside or inside) .... outside
Selection[1] (roughing with or without rounding) .... with rounding
Infeed depth (depth for one rough cut) 3
Finishing allowance in Z 0.1
Finishing allowance in X (radial input) 1.0
Finishing allowance contour paralel 0.1
Roughing feedrate 0.4
Pluging feedrate (for undercuts) 0.1
Finishing feedrate 0.15
Dwell time for chip breaking 0
Path length for chip breaking 0
Retraction path (for tool retract)... 1.0

[1] Selection with softkey ALTERNATIV or with the key ⟳.

Kontur "CONTOUR1"
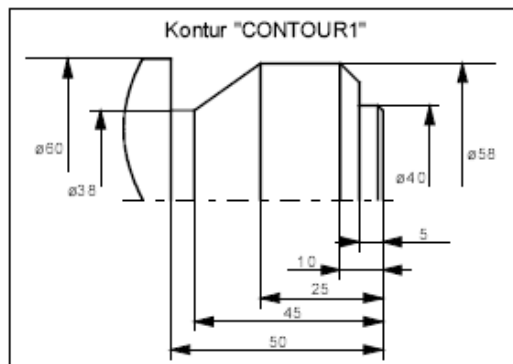
Longitudinal turning - external contour

```
G54
G53 G0 D0 X... Z... (Z2=...)
N1 T1 D1 ....................................... (copying steel)
G96 S250 M4
G0 X65 Z0
G1 F0,18 X-1,6

;NCG#CYC95#turning.com#NC1#*NCG;*RO*
;*HD*;#7#"complete"#"long"#"outside"#....
CYCLE95("CONTOUR1",3,0.1,1,0.1,0.4,0.1,0.15,
        9*,0,0,1)
;#END#*NCG;*RO*;*DH*

Subroutine: CONTOUR1:

G1 X35 Z2
Z0
X40 Z-1
Z-5
X50
X58 Z-10
Z-25
X38 Z-45
Z-50
X60 CHR=0,3
Z-50,4
M17
```

*Value for machining type (see table type of machining

22
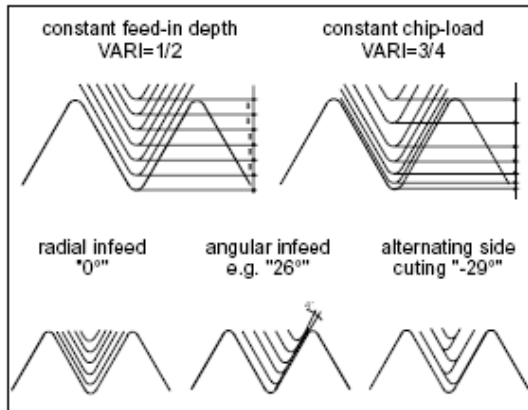
infeed possibilities for "CYCLE97"

Threading Cycle "CYCLE97"

It is possible to make cylindrical or tapered, external or internal threads with constant pitch, longitudinal or transversal. The threads can be a "single start" or a "multiple start" thread.

You can choose between two different ways of feed-in: constant feed-in depth or constant chip-load.

value	thread	infeed
1	outside	constant infeed
2	inside	constant infeed
3	outside	constant chip section
4	inside	constant chip section

Note:
Further informations see Siemens-book "Cycles"

Numer of roughing passes insteel accoring to Sandvik
external / internal (metric. threads 60°)

pitch	number of cuts	pitch	number of cuts
0,5	4	3	12
0,75	4	3,5	12
1	5	4	14
1,25	6	4,5	14
1,5	6	5	14
1,75	8	5,5	16
2	8	6	16
2,5	10		

Example - external thread
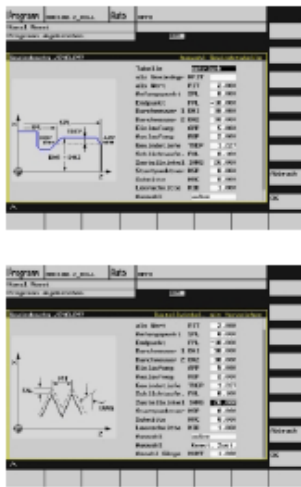
Table[2] (metric or none) .... metric
Input as thread size[2] 0
Thread pitch as value 2
Start point of the thread in the longitudinal axis 0
End point of the thread in the longitudinal axis .... -10
Diameter 1 of the thread at the start point 30
Diameter 2 of the thread at the end point 30
Run-in path, without sign 5
Run out path, without sign 2
Thread depth, without sign 1.227
Finishcut allowance, without sign 0.1
Infeed angle, with sign (" -" ...alternate infeed) 26
Start point offset (first thread start) in degrees 0
Roughing passes (number of cuts) 8
Idle passes (number of non-cuts) 1
Selection[1] of thread tpye (outside or inside) .... outside
Selection[1] machining type
(const. infeed or const. chip load section) .... const. infeed
Start of threads (number of threads) 1
Retraction path (for tool retreat) 0,5

[1] Selection with softkey ALTERNATIV or with the key .

[2] Thread pitch and thread depth are entered automatically with input of the thread size for metric standard threads

```
G54
G53 G0 D0 X... Z... (Z2=...)
N2 T2 D1                          ;Threading tool, pitch 2mm
G95 S2000 M3

;NCG#CYC97#turning.com#NC1#*NCG;*RO*;*HD*;#metric#....
#"external"#"const.infeed"#....#*NCG;*RO*;*HD*
CYCLE97(2,,0,-10,30,30,5,2,1:227,0:1,26,0,8,1,1,1)
;#END#*NCG;*RO*;*HD*

G0 X200 Z100
M30
```
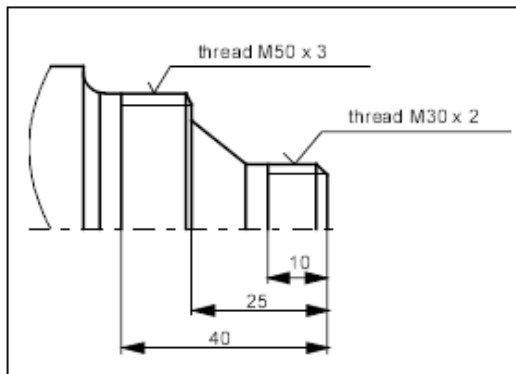

external thread

Calculation of radial depth of thread "external thread - 60°"

$$t = 0{,}6134 \times p$$

t .... thread depth
p .... pitch

Example - internal thread

Table[2] (metric or none) .... metric
Input as thread size[2] 40
Thread pitch as value 4
Start point of the thread in the longitudinal axis 0
End point of the thread in the longitudinal axis .... -20
Diameter 1 of the thread at the start point.......................... 35.67
Diameter 2 of the thread at the end point.................................. 35.67
Run-in path, without sign 5
Run-out path, without sign 1
Thread depth, without sign 2.165
Finishcut allowance, without sign...0
Infeed angle, with sign (" -" ...alternate infeed) .... -29
Start point offset (first thread start) in degrees 0
Roughing passes (number of cuts) 14
Idle passes (number of non-cuts) 0
Selection[1] of thread tpye (outside or inside) .... inside
Selection[1] machining type
(const. infeed or const. chip load section)...................... const. infeed
Start of threads (number of threads) 1
Retraction path (for tool retreat) 0,2

[1] Selection with softkey ALTERNATIV or with the key 🔵.

[2] Thread pitch and thread depth are entered automatically with input of the thread size for metric standard threads

```
G54
G53 G0 D0 X... Z... (Z2=...)
N4 T5 D1                           ;Threading bar, pitch 4mm
G95 S2000 M3

;NCG#CYC97#turning.com#NC1#*NCG;*RO*;*HD*;#metric#....
#"internal"#"const X-sect."#....#*NCG;*RO*;*HD*
CYCLE97(4,,0,-20,35.67,35.67,5,1,2.165,0,-29,0,14,0,4,1)
;#END#*NCG;*RO*;*HD*

G0 X200 Z100
M30
```
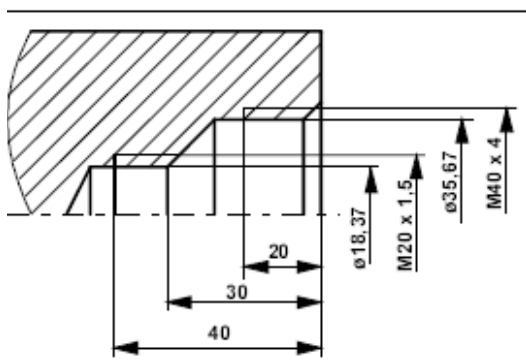
Calculation of radial depth of thread "internal thread - 60°"

$$t = 0,5413 \times p$$

t .... thread depth
p .... pitch

Cycles of the "User" group

Deep Hole Drilling Cycle "CYCLE83E"

This cycle is used for drilling of deep holes, either in X-axis or z-axis direction.
Also you can choose between CHIP-BREAKING or SWARF-REMOVAL.

The cycle was optimized by EMCO for the following reasons:

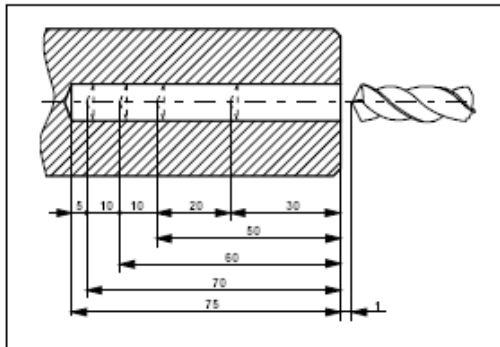• Drilling is possible in the turning plane G18. Thus, a permanent change of the working plane can be avoided.
• The drills can be declared as turning tools (tool type 500, L1=X, L2=Z)

> **Note:**
> Further informations see Siemens-book "Cycles"

Reference plane, absolut 1
Final drilling depth, absolut .... -75
First drill depth, absolut .... -30
Degression[1] for depth reduction, without sign 10
Dwell at end point (sec) 0
Dwell at start point (sec) 0
Selection[1] of operation type:
Chip breaking or swarf removal ..... chip breaking
Selection[1] drilling axis: X - axis or Z - axis .... Z

[1] Selection with softkey ALTERNATIV or with the key.
[2] Any further drilling depth is reduced by the degression value (minimum drilling depth = degression value)

Example - Drilling with the counter spindle

(Without driven tools. Only possible in coordinate X0)

```
G55          ;set reference point for counter
             spindle side
MIRROR Z0    ;mirroring the Z axis around G55 Z0
             for normal programming on the
             counter spindle side
SETMS(2)     ;setting the counter spindle as
             master spindle
G53 G0 D0 X... Z... (Z2=...)
N8 T8 D2
G95 S2000 M4 F0,1
G0 X0 Z5

CYCLE83E(1,-75,-30,10,0,0,1,1)

G0 X200 Z10
M30
```

Example - Drilling with the main spindle

(Without driven tools. Only possible in coordinate X0)

```
G54
G53 G0 D0 X... Z... (Z2=...)
N8 T8 D1
G95 S2000 M3 F0,1
G0 X0 Z2

CYCLE83E(1,-75,-30,10,0,0,1,1)

G0 X200 Z10
M30
```

Example
drilling with driven tools (axial):

## Example
drilling with driven tools (radial):

Reference plane, absolut 1
Final drilling depth, absolut .... -30
First drill depth, absolut .... -15
Degression[2] depth reduction, without sign 3
Dwell at end point (sec) 0
Dwell at start point (sec) 0
Selection[1] of operation type .... Swarf removal
Selection[1] drilling axis (X=0 or Z=1) .... Z

Reference plane, absolut 62
Final drilling depth, absolut .... -5
First drill depth, absolut 40
Degression[2] depth reduction, without sign 5
Dwell at end point (sec) 0
Dwell at start point (sec) 0
Selection[1] of operation type .... Swarf removal
Selection[1] drilling axis (X=0 or Z=1) X

[1] Selection with softkey ALTERNATIV or with the key
[2] Any further drilling depth is reduced by the degression value (minimum drilling depth = degression value)
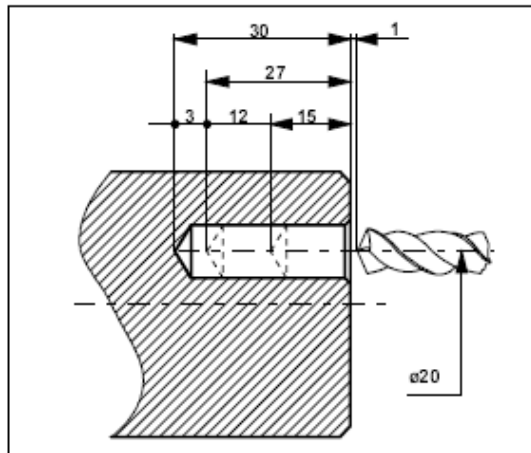
axial drilling with driven tools

```
G54
G53 G0 X... Z... (Z2=...)
N8 T8 D1 ;          drill diam. 6mm - driven
SETMS(3) ;          set driven tools as MASTER-
                    SPINDLE
G95 S1000 M4
SPOS[1]=0 ;         turn on C-axis and position at
                    0°
G0 X20 Z2 F0,12

CYCLE83E(1,-30,-15,3,0,0,1,1)

G0 X200 Z10 M5
SETMS(1) ;          set main spindle as MASTER-
                    SPINDLE
M5
M30
```

radial drilling with driven tools

```
G54
G53 G0 X... Z... (Z2=...)
N8 T8 D1 ;          drill diam. 6mm - driven
SETMS(3) ;          set driven tools as MASTER-
                    SPINDLE
G95 S1000 M3
SPOS[1]=0 ;         turn on C-axis and position at
                    0°
G0 X70 Z-20 F0,12

CYCLE83E(62,-5,40,-5,0,0,1,0)

G0 X200 Z10 M5
SETMS(1) ;          set main spindle as MASTER-
                    SPINDLE
M5
M30
```
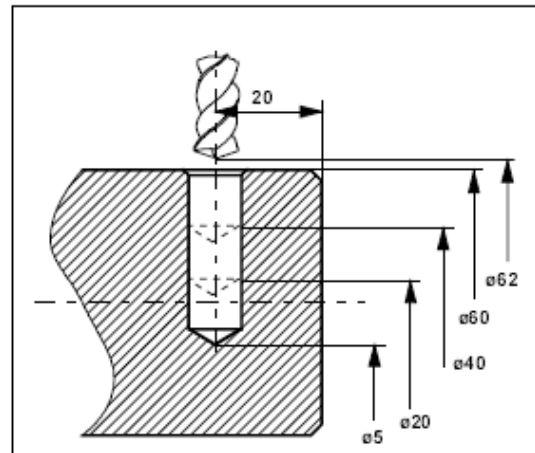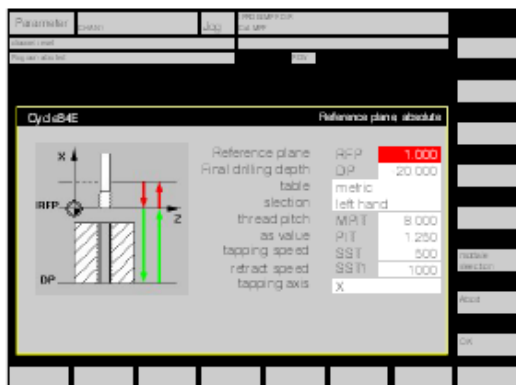
axial tapping on main- or counterspindle

Tapping "CYCLE84E"

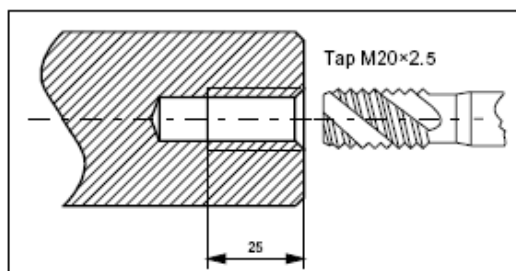**The cycle was optimized by EMCO for the following reasons:**

- Drilling is possible in the turning plane G18. Thus, a permanent change of the working plane can be avoided.
- The taps can be declared as turning tools (tool type 500, L1=X, L2=Z)

> **Note:**
> Further informations see Siemens-book "Cycles"

Axial at the Main spindle or Counter spindle without Floating holder

Referenz plane, absolute	3
Final drilling depth, absolute	-25
Table[2] (metric or without table)	metric
Selection[1] sense of rotation (M3/M4)	M3
Selection[1][2] thread nominal diameter	20
Thread pitch as value	2,5
Tapping speed (rpm)	600
Retract speed (rpm)	800
Selection[1] tapping axis (X=0 or Z=1)	Z

[1] Selection with softkey ALTERNATIV or with the key.

[2] automatic input of the thread pitch during input of the nominal thread diameter for metric standard threads

With the main spindle
(Without driven tools. Only possible in coordinate X0)
```
G54
G53 G0 D0 X... Z... (Z2=...)
N8 T8 D1     ;Tap M20×2.5, clamped fast
G0 X0 Z5 M5

CYCLE84E(3,-25,(-)2.5,600,800,1)
                ;"-" before pitch= counterclockwise
                   rotation (M4)
G0 X200 Z10
M30
```

With the counter spindle
(Without driven tools. Only possible in coordinate X0)

```
STOPRE       ; read-in stop until actual block
G55          ;set reference point for counter
               spindle side
MIRROR Z0    ;mirroring the Z axis around G55 Z0
               for normal programming on the
               counter spindle side
SETMS(2)     ;setting the counter spindle as
               master spindle
G53 G0 D0 X... Z... (Z2=...)
N8 T8 D2     ;taps M20×2.5, clamped fast
G0 X0 Z5

CYCLE84E(3,-25,(-)2.5,600,800,1)
                ;"-" before pitch= counterclockwise
                   rotation (M4)
G0 X200 Z10
M30
```

Axial with driven tools without floating holder
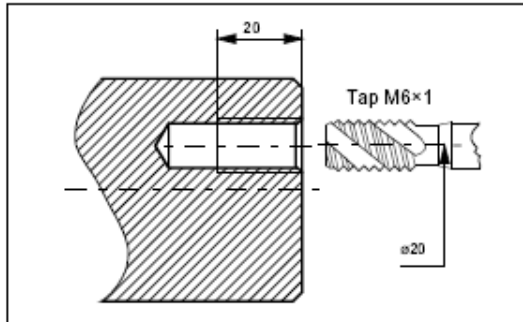
Radial with driven tools without floating holder

Note:
The sense of rotation of the driven tools depends on the construction of the axial and radial toolholders!
Therefore, the sense of rotation must be checked before in any case!
A wrong translation of the sense of rotation due to the angle of the toolholder can only be compensated for with the change of the selection of the thread type (right- or lefthanded thread).
This cycle can be called modal for further tapping position by pushing the softkey MODAL CALL.

It can be bored with traditional bore- and mill-holders axial or radial.

axial tapping with driven tools (without floating holder)

Reference plane, absolute 2
Final drilling depth, absolute .... -20
Table[1] (metric or none)) .... metric
Selection sense of
rotation (M3/M4), **see note** M3
Thread nominal diameter[1] 6
Thread pitch as value 1
Tapping speed (rpm) 1000
Retract speed (rpm) 2000
Selection tapping axis (X=0 or Z=1) .... Z

1) automatic input of the thread pitch during input of the nominal thread diameter for metric standard threads

```
G54
G53 G0 D0 X... Z... (Z2=...)
N8 T8 D1      ;taps M6×1, clamped fast
SETMS(3)      ;setting the tool spindle as master
              spindle
G95 G0 X20 Z2
SPOS[1]=0     ;turn on C-axis and position at 0°

CYCLE84E(2,-20,(-)1,1000,2000,1)
              ;"-" before pitch= counterclockwise
              rotation (M4)

G0 X200 Z10 M5
SETMS(1)      ;set main spindle as MASTER-
              SPINDLE
M5
M30
```
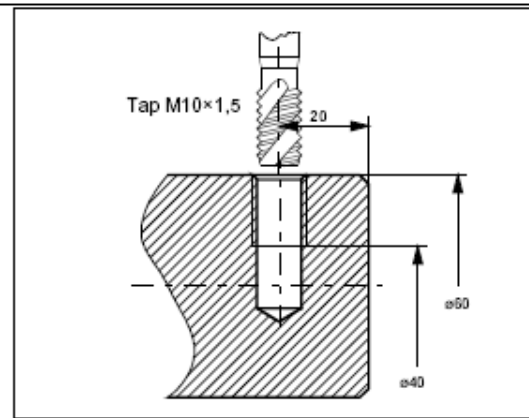

radial tapping with driven tools (without floating holder)

Reference plane, absolute 65
Final drilling depth, absolute 40
Table[1] (metric or none) .... metric
Selection sense of
rotation (M3/M4), **see note** M3
Thread nominal diameter[1] 10
Thread pitch as value 1.5
Tapping speed (rpm) 1000
Retract speed (rpm) 2000
Selection tapping axis (X=0 or Z=1) X

1) automatic input of the thread pitch during input of the nominal thread diameter for metric standard threads

```
G54
G53 G0 D0 X... Z... (Z2=...)
N8 T8 D1      ;taps M10×1.5, clamped fast
SETMS(3)      ;setting the tool spindle as master
              spindle
G95 G0 X70 Z-20
SPOS[1]=0     ;...turn on C-axis and position at 0°

CYCLE84E(65,40,(-)1.5,1000,2000,0)
              ;"-" before pitch= counterclockwise
              rotation (M4)

G0 X200 Z10 M5
SETMS(1)      ;set main spindle as MASTER-
              SPINDLE
M5
M30
```

Programming the automatic tailstock

here you will find two different cases of applications.

- tailstock suports the shaft at one length position
- tailstock suports the shaft at two or more different length position (used for multiple bar feed out)

Supporting on one length position

- Clamp the shaft into the mainspindle just as you would machine it.
- Move tailstock towards the part in JOG mode.
- Then the preasure builds up and the machine memorizes this position.
- Move tailstock back into home-position

in program:
```
....
M5
M21   ; Tailstock forward
N2 T2 D1 G96 S200 M4
....      ; machining
M5
M20   ; tailstock back to home position
M30
```

Supporting on more length positions

- Clamp the shaft into the mainspindle just as you would machine it.
- Move tailstock towards the part in JOG mode until preasure builds up.
- Press Softkey "Diagnosis"
- Press Softkey "PLC"
- Press Softkey "Read in Operand"
- Set cursor to "TAILSTOCK.PLC" an press Softkey "OK".
- The listened value in the table (in format "d") is the position to be programmed.

1 unit = 5 mm = 1u
100mm
≈325mm
11u
Home position
= (95-10) * 5=425mm
95u
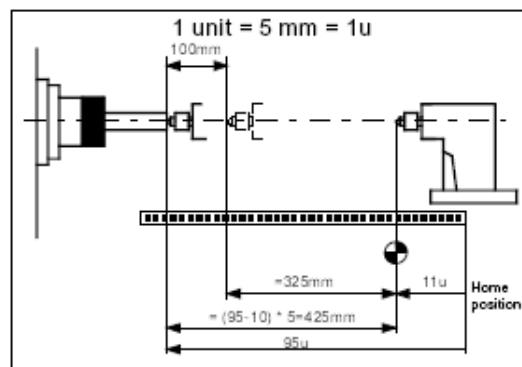
in program:
```
....
M5
M21 H425              ; Tailstock forward
N2 T2 D1 G96 S200 M4
....                  ; machining
M5
M20  H220             ; retract tailstock 200 mm (to
                        the right)
... bar feed out 100 mm
M21 H325              ; Tailstock forward 100 mm
                        (to the left)
.... machining
M20                   ; tailstock back to home
                        position
M30
```
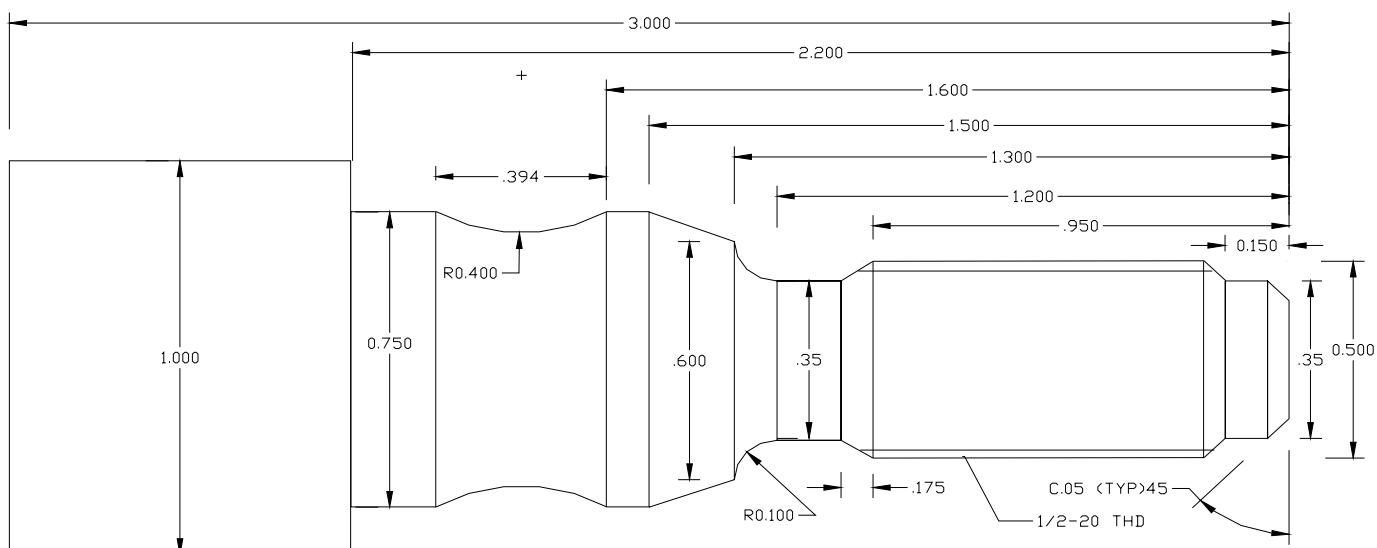
Demo 1
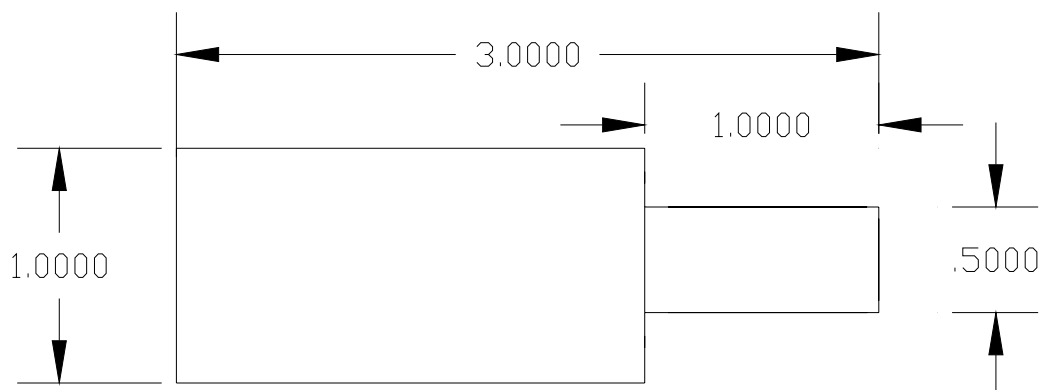
Main

```
G54      ; SIDE 1 OFFSET- SET IT AND SAVE IT  8.2687
SETMS(1) ; MAIN SPINDLE TO #1 SPINDLE, 2=COUNTER SPINDLE 3=TURRET
;BAR1    ; RUN BAR1 SUB-PROGRAM,  CHECKS FOR NEW BAR TO BE LOADED
LIMS=3800 ; SET MAX RPM
G0 G53 X8.0 Z15.0 D0   ; RAPID MOVE TO SAFE INDEX POSITION
;GOTOF AAA ; JUMP STATMENT - USED FOR SET-UP OF NEW PROGRAM
;****************************************************************
;
T1 D1 G96 S900 M4  ; ROUGH FACE AND PART O.D.
G0 X1.75 Z.1
M103 ; MAKES SURE THE SPINDLE IS WITHIN 10% OF REQUIRED RPM
CYCLE95("OD1",0.1,0.003,0.015,0.0,0.01,0.006,0.004,1,0,0,0.08) ; ROUGHING CYCLE
G0 X2.0 Z4.0 M5 M9   ; SUB-PROGRAM "OD1" HAS THE FINISHED PART PROFILE
;G0 G53 X8. Z15. D0 ; SAFE INDEX POINT
M30
```

OD1

```
G0 X0
G1 Z0 F.005
X.5
Z-.1
X1.75
M17
```

DEMO 2

MAIN
MSG(DEMO2)
G54   ; SIDE 1 OFFSET- SET IT AND SAVE IT 8.2687
SETMS(1) ; MAIN SPINDLE TO #1 SPINDLE, 2=COUNTER SPINDLE 3=TURRET
; BAR1   ; RUN BAR1 SUB-PROGRAM, CHECKS FOR NEW BAR TO BE LOADED
LIMS=3800 ; SET MAX RPM
G0 G53 X8.0 Z15.0 D0   ; RAPID MOVE TO SAFE INDEX POSITION
; GOTOF AAA ; JUMP STATMENT - USED FOR SET-UP OF NEW PROGRAM
;***
;
T1 D1 G96 S900 M4 ; ROUGH FACE AND PART O.D.
MSG("55 DEG. O.D.TOOL .015 TNR")
G0 X2.0 Z.1 ; M8
M103 ; WAIT FOR RPM TO BE REACHED
G1 Z0.0 F.015
X-.04 F.008
G0 X1.75 Z.05
M103 ; MAKES SURE THE SPINDLE IS WITHIN 10% OF REQUIRED RPM
CYCLE95("OD1",0.1,0.003,0.015,0.0,0.01,0.006,0.004,9,0,0,0.08) ; ROUGHING CYCLE
G0 X2.0 Z4.0 M5 M9   ; SUB-PROGRAM "OD2" HAS THE FINISHED PART PROFILE
;G0 G53 X8. Z15. D0 ; SAFE INDEX POINT
M30

OD1
G0 G42 X.2   X.75 Z-1.5
G1 Z0   Z-1.6
X.35 CHR=.05   G2 X.75 Z-1.994 CR=.4
Z-.15   G1 Z-2.2
X.5 CHR=.05   G1 X1.75
Z-.950   M17
X.35 Z-1.125
Z-1.3 RND=.1
X.6

PROGRAM INFORMATION PAGE

BEGINNING OF A EXPORTED PROGRAM

```
;%_N_MAIN_PROGRAM_MPF        ; PROGRAM DIRECTORY
;$PATH=/_N_WKS_DIR/_N_DEMO1_WPD ; PATH FOR THE DIRECTORY
   ; PART NAME 123456 REV.A  -  TURN AND MILL COMPLETE
   ; 2.0 DIA. STOCK 1.5 OUT FROM COLLET
;*************************************************************
;
```

HAVING WORK SHIFTS CALL OUT IN PROGRAM

```
;%_N_WORKSHIFTS_SPF
;$PATH=/_N_WKS_DIR/_N_DEMO1_WPD

$P_UIFR[1]=CTRANS(Z,8.750)  ; LOAD G54 WORKSHIFT
$P_UIFR[2]=CTRANS(Z,30.162) ; LOAD G55 WORKSHIFT
$P_UIFR[3]=CTRANS(Z,30.162) ; LOAD G56 WORKSHIFT
```

SUB PROGRAM DIRECTORY

```
%_N_NAME_SPF
;$PATH=/_N_SPF_DIR
```

BAR LOAD

```
%_N_BAR1_SPF
;$PATH=/_N_SPF_DIR
IF $A_IN[38]==0 GOTOF NOEND ; LOOKING FOR END OF BAR SIGNAL
M67
M25 ; OPEN CHUCK
M1=25
M57 ; SPINDLE OSCILLATION ON
M69 ; BAR CHANGE
NEWBAR: ;LABLE FOR THE LOOP
MSG ("CHANGE OF BAR")
IF $A_IN[39]==1 GOTOB NEWBAR ;  WAIT FOR NEW BAR CHANGE
MSG ("CUT OFF END")
M26 ; CLOSE CHUCK
M1=28
M58 ; SPINDLE OSCILLATION OFF
STOPRE
```

```
M68
SETMS(1)
LIMS=3500
T12 D1
G96 S350 M4
G0 G90 Z.02
X1.12 ;    **** THIS MUST BE CHANGED FOR STOCK SIZE **** ;
M103
G1 X.2 F.002 M8
G97 S1500 ; SLOW DOWN TO DROP SLUG
X-.02 F.001
G0 X1.5 M5
G4 F1
G0 G53 X5.9 Z8 D0 M9 ; SAFE INDEX POSITION
NOEND:
RET
```

TMC (MILLING WITH LIVE TOOLS) IN G17 EXAMPLE

```
M5 ; YOU MUST STOP MAIN SPINDLE BEFOR LIVE TOOL - C-AXIS

SETMS(2) ; SETS THE MAIN SPINDLE TO #2 ( SPINDLE IN THE TURRET )
TMCON    ; TURNS ON THE MILLING MODE, X Y CORD. G17
PROGRAM
M5 ; MUST STOP LIVE TOOLING FOR INDEXING
TMCOFF ; TURNS OFF THE MILLING MODE
STOPRE ; THIS IS SO THE CONTROLL WILL NOT LOOK AHEAD - STOP
READ -
```

C AXIS ROTATE EXAMPLE

```
M5
SETMS(2)
S3800 M4 ;LIVE TOOL
SPOS[1]=45 ; TURNS ON THE C-AXIS MODE AND POSITIONS AT 45DEG.
C45      ; POSITION  C-AXIS AT 45 DEGREE - REPETE YOUR FIRST
POSITION

M5
```
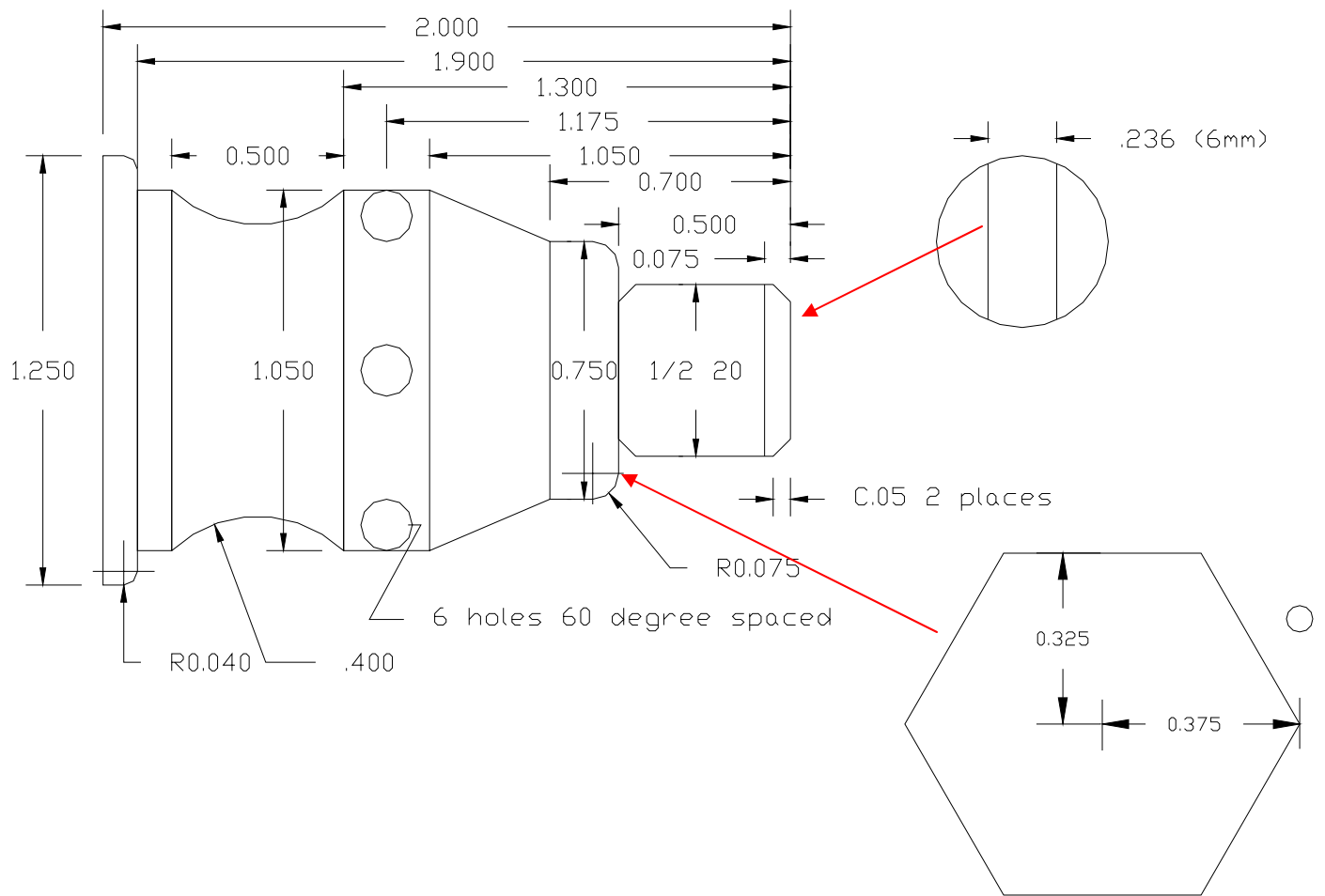
HEX INFORMATION

```
%_N_HEXL_SPF
;$PATH=/_N_SPF_DIR
PROC HEXL (REAL SW,REAL RAD)SAVE
R1=SW/2
R2=COS(30)
R3=TAN(30)
R4=R1/R2
R5=R1*R3
G0 X=R4+RAD+0.1 Y0
G1 G42 X=R4 Y0
X=R5 Y=R1
X=-R5 Y=R1
X=-R4 Y0
X=-R5 Y=-R1
X=R5 Y=-R1
X=R4 Y0
Y0.12
G0 G40 X=R4+RAD+0.08 Y0
RET

%_N_HEXR_SPF
;$PATH=/_N_SPF_DIR
PROC HEXR (REAL SW,REAL RAD)SAVE
R1=SW/2
R2=COS(30)
R3=TAN(30)
R4=R1/R2
R5=R1*R3
G0 X=R4+RAD+0.1 Y0
G1 G41 X=R4 Y0
X=R5 Y=-R1
X=-R5 Y=-R1
X=-R4 Y=0
X=-R5 Y=R1
X=R5 Y=R1
X=R4 Y0
Y-0.12
G0 G40 X=R4+RAD+0.08 Y0
RET

%_N_SQUARE_SPF
;$PATH=/_N_SPF_DIR
PROC SQUARE (REAL SW,REAL RAD)SAVE
R1=SW/2
R2=R1*1.4142136
G0 X=R2+RAD+0.1 Y0
G1 G41 X=R2 Y0
X=0 Y=-R2
X=-R2 Y0
X=0 Y=R2
X=R2 Y0
Y-0.12
G0 G40 X=R2+RAD+0.1 Y0
RET
```

DEMO 3

2.000

1.900

1.300

1.175

1.050

0.700

0.500

0.075

0.500

1.250

0.500

1.050

0.750

1/2 20

.236 (6mm)

C.05 2 places

R0.075

6 holes 60 degree spaced

R0.040   .400

0.325

0.375

MAIN
EXTERN HEXR (REAL,REAL)
MSG(DEMO3)
G54
SETMS(1)
LIMS=3800
PARK
; GOTOF AAA
.***
;
M1
T2 D1 G96 S900 M4
MSG("80 DEG. LH O.D.TOOL .032 TNR")
G0 X2.0 Z.1 ; M8
M103
G1 Z0.003 F.015
X-.04 F.008
G0 X1.75 Z.05
M103
CYCLE95("OD1",0.1,0.003,0.015,0.0,0.01,0.006,0.004,1,0,0,0.08) ; ROUGHING CYCLE
PARK
.***
;
;AAA:
M1
T1 D1 G96 S900 M4
MSG("55 DEG. LH O.D.TOOL .015 TNR")
G0 X1.75 Z.05 ; M8
CYCLE95("OD1",0.1,0.003,0.015,0.0,0.01,0.006,0.004,5,0,0,0.08) ; FINISH CYCLE
PARK
.***
;
;AAA:
M1
T1 D1 G97 S1000 M3
MSG("60 DEG O.D. RH THREADING TOOL")
G0 X.55 Z.1 ; M8
CYCLE97(0.05,0,-.460,0.394,0.394,0.12,0,0.033,0.001,29.5,0,4,1,3,1)
PARK
.***
;
;AAA:
M1
M5
SETMS(2)
STOPRE
TMCON
T6 D1
MSG("3/8 E.M. LIVE AXIAL HOLDER ")
G95 S900 M4 F0.12
X1.0 Z.1
G1 Z-.7
HEXR(.625,.187) ; A/F - .625, radius of mill - .187
G0 X.900 Y0 M5
TMCOFF
PARK
M30