SIEMENS

SINUMERIK 810/840D
DIN Programming for Milling

Training Manual Edition 2008.01

Training Documentation

SIEMENS

SINUMERIK 810/840D

Operating and Programming
DIN - Milling

Valid for:

Control
SINUMERIK 810/840D

Edition 01.2008

Module content

Operating and Programming - Milling

A301	Technoloy Basics
A302	Geometry Basics
A303	Simple Contour elements
A304	Mathematical Principles
A305	Zero offset and reference points
A309	Program structure
A310	Contour milling
A311	Programming of Subroutines - Milling
A312	Jumps - Repeats - Milling
A313	Circular and square pockets
A314	Mirror - Offsets - Rotation - Scale - Milling
A315	Working with Cycles
A316	Milling Cycles
A317	Drilling Cycles - Milling
A340	Operation_840D

A301

General technological aspects

1 Brief description

Aim of the module:

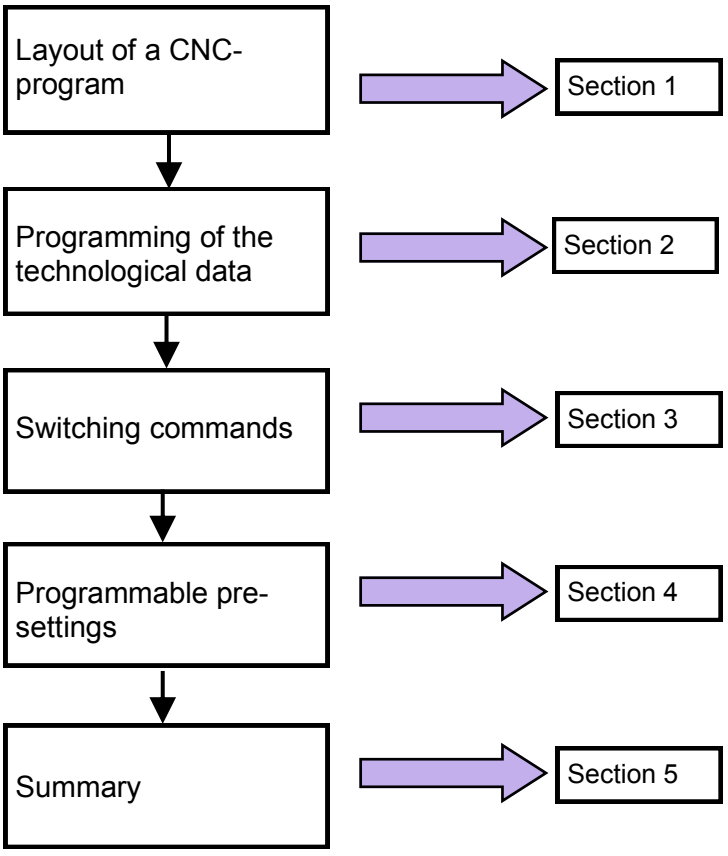
Having worked through this module you will be familiar with the most important technological aspects and machine functions.

Description of the module:

This module shows the general layout of a program with respect to the technological commands as per DIN 66025-2 for Turning and Milling.

Content of the module:

- Layout of a CNC-program
- Programming of the technological data
- Switching commands
- Programmable pre-settings
- Summary



Notes

A **CNC-program**, also known as Part program, consists of a logical sequence of commands, which are executed step-by-step by the control unit after the program has been started.
The manufacturers of control units recognize and apply the guidelines as per DIN 66025-2.

Each program is compiled and stored under a **program name** in the control unit. The name can contain letters as well as numbers.

A block starts with a block number followed by the commands.

Each command consists of command words, which in turn consist of an **address letter** (A-Z) and an associated **numerical value.** (Both upper or lower case characters are permissible)

Program layout:

Block Nr..	Departure information							Switching information			
	Auxiliary command	Co-ordinate axes			Interpolation parameter			Feed	Speed	Tool	Misc. function
N	G	X	Y	Z	I	J	K	F	S	T	M

Geometrical data — Technological data

The block number is a program-technical assignment, which is not evaluated by the control unit as a command. It is usually programmed to go up in steps of 10 and serves only the user for better oversight. It has no effect on the program execution.

The geometrical data include all instructions that clearly define mathematically the motion of the tool or the axes.

The technological data are used for instance to activate the required tool and to pre-select the necessary cutting parameters feed rate and spindle speed. Miscellaneous functions can control for example such things as direction of rotation and auxiliary appliances.

Programming example:

```
....
N80   T1; Roughing tool
N90   M6
N100 G54 F0.2 S180 M4
N110 G00 X20 Y0 Z2 D1
N120 ……  …..
```

In order to improve the oversight within a program, commentaries can be optionally added at the end of a block. These must be preceded by a semicolon; Any characters that follow thereafter will not be taken account of by the control unit..

Notes

Before every technological working step in a CNC-Program the respective tool must be selected by means of the addresses **T** and **D**.

The address T is followed by the name of the tool, which may be stated either with numbers or letters. (Here only the variant using numbers will be dealt with.)

All applicable tool data (e.g. tool type, length, radius etc.) are activated in the program with the address D.
Here a complete set of data D is referred to as „Cutting edge“.

Several cutting edge numbers (D1 … D9) may be generated for each tool.

Programming example:	*Explanation:*
N10 T17 ; Drill	*Block 10, call-up of tool 17, Commentary to the tool*
N20 M6	*Tool change,*
N30 … D1	*The cutting edge D... must be activated in the block with the first departure com-- mand*

After the call-up of the tool follows the infinitely variable selection of the optimum cutting values with the addresses **F** and **S.**

The feed rate with the address F can be entered either as feed per min v_f (in mm/min) or as feed per revolution f (in mm).

Generally speaking the following starting status for the machines applies:

- Milling machines with feed rate v_f in mm/min
- Code **G94**

- Turning machines with feed per revolution f in mm
- Code **G95**

Programming example 1:	*Explanation:*
N10 T20 ; Endmill	
N20 M6	
N30 G94 F200 S1000 M3 D1	*v_f = 200m/min, n = 1000 min^{-1}*
N40 ….	

Programming example 2:	*Explanation:*
N10 T2; Turning tool, finishing	
N20 G96 F0.1 S200 M4 D1	*f = 0,1mm, v_c = 200 m/min*
N30 ….	

Switching commands

The preset motion of the work spindle in the respective direction of rotation is started with the respective switching commands.
Further additional functions can control, for instance, cooling circuits, clamping means, auxiliary functions and the running of the program.

Since the number of additional functions depends entirely on the constructive and technological equipment of the machine, the following list should be seen only as an extract of possible instructions.

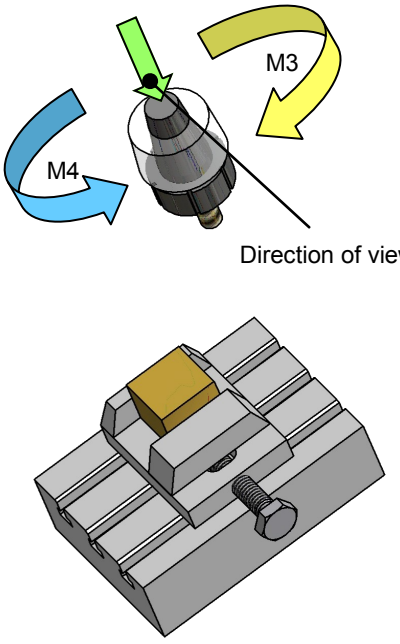
Instruction	Meaning
M00	Programmed Halt
M03	Work spindle ON, clockwise
M04	Work spindle ON, anti-clockwise
M05	Work spindle Halt (however, the program continues)
M06	Tool change
M08	Coolant ON
M09	Coolant OFF
M30	End of program; jump back to the start of the program

Programming example:	*Explanation:*
N10 T1; Face mill	
N20 M6	*Tool change*
N30 G94 G97 F600 S2500 D1	*v_f = 600m/min, n = 2500 min^{-1}*
N40 M3 M8	*Spindle ON clockwise, coolant ON*
…..	
N90 M30	*End of program*

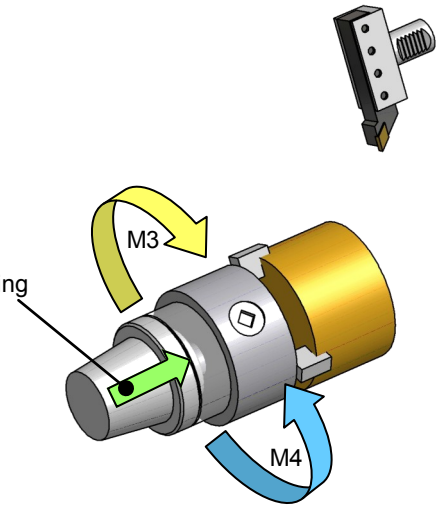
(Further functions will be found in the annex to this manual)

Effect of the switching commands M3 and M4

Example Milling



Example Turning



Notes

When starting a part program the basic settings as defined by the manufacturer will be activated. These depend on the individual machine specification and apply thereafter for the whole of the program run (modal) unless they are changed by the operator by programming.

This section describes just a few of the multitude of possible selections for turning and milling machines that deserve highlighting.
(Codes that have already been dealt with are no longer included)

Continuous path behaviour:

Exact stop **Code G9** block-by-block **Code G60** * modal

In order to reach the final position precisely the path velocity is reduced at the end of the block towards zero. This is useful, for instance, to obtain relatively sharp edges when machining around contour corners.
However, it must be borne in mind that, if there are too many positioning sequences, the additional machining time required in consequence cannot be altogether disregarded.

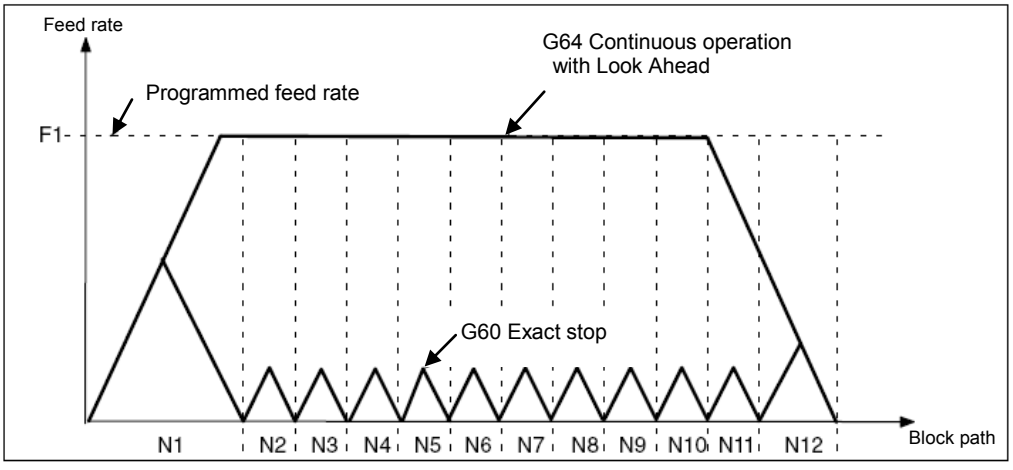
Continuous control operation Code G64

In this case the tool moves as much as possible with constant velocity without deceleration at the end of a block. Hence the machining time is less than under the continuous path status „Exact stop“.

The corners of contours are machined without any relief and therefore the corners are not so sharply defined.
With this function the control works with a speed control taking into account several blocks ahead (Look Ahead).

The even speed in this instance results in better cutting conditions and also a better surface quality.

The following image compares the frequent braking and accelerating sequences between the individual blocks in case of G60 and the constant speed in case of G64.



* Usual preset starting status