M551

# General technology basics

## 1 Brief description

**Objective of the module:**

Working through this module you become familiar with the most important technological aspects and machine functions.

**Description of the module:**

This module explains the general layout of a program, with respect to the technological commands as per DIN 66025 for turning and milling.

**Content:**

Layout of a CNC-program

Programming of the technology data

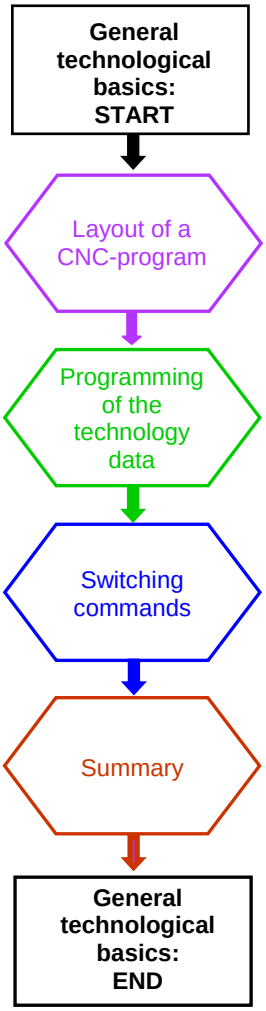
Switching commands

Summary

**828D/840Dsl SINUMERIK Operate**

## General geometry basics: Description

This module explains the general layout of a program, with respect to the technological commands as per DIN 66025 for milling.




Notes

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.

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 | Coordinate 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 the user only for a 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 overview within a program, comments 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 T="DRILL_12 mm" | *Block 10, call-up of tool* |
| N20 M6 | *Block 20, Tool change,* |
| N30 … D1 | *The cutting edge D... must be activated in the block with the first axis movement.* |

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

The feed rate $\boldsymbol{v_f}$ with the address "**F**" can be entered either as feed per min (in mm/min) or as feed per revolution (in mm/rev).

The Cutting speed $\boldsymbol{v_c}$ with the address "**S**" can be entered either as spindle speed in revolutions per minute (rev/min) or direct as cutting speed in meters per minute (m/min).

Default status of the machine when they are powered up are as follows:

- Milling machines with feed rate "**F**" in mm/min — Code **G94**

- Deactivate constant cutting speed "**S**" — Code **G97**
  spindle speed in rev/min (default)

| **Programming example:** | ***Explanation:*** |
|---|---|
| N10 T="HM_SF20mm" | |
| N20 M6 | |
| N30 G94 F200 S1000 M3 D1 | $v_f$ *= 200 mm/min, n = 1000 min$^{-1}$* |
| N40 …. | |

Notes

There are different commands to control the direction of rotation of the work spindle.
Additional auxiliary functions can for example control cooling circuits, clamping devices, auxiliary functions and running of the program.
But the presence of these additional functions depends entirely on the technology and the machine design.
The following list should be only considered as an example of commands:

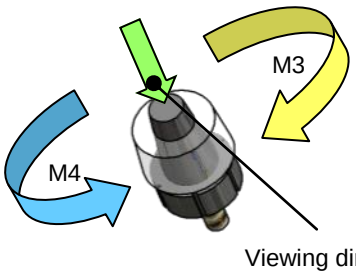
| Instruction | Meaning |
|---|---|
| **M00** | Programmed Stop |
| **M03** | Work spindle ON, clockwise |
| **M04** | Work spindle ON, anti-clockwise |
| **M05** | Work spindle Stop (however, the program continues) |
| **M06** | Tool change |
| **M08** | Coolant ON |
| **M09** | Coolant OFF |
| **M30**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

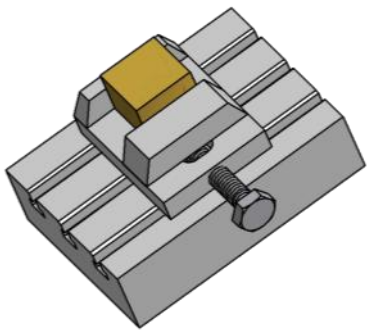
| **Programming example:** | ***Explanation:*** |
|---|---|
| N10 T1="Face mill" | |
| N20 M6 | *Tool change* |
| N30 G94 G97 F600 S2500 D1 | $v_f$ *= 600 mm/min, n = 2500 min$^{-1}$* |
| N40 M3 M8 | *Spindle ON clockwise, coolant ON* |
| ….. | |
| N90 M30 | *End of program* |

(**Note:** Further functions can be found in the annexure of this manual)

**Effect of the switching commands M3 and M4**

Example Milling



| Address | Meaning |
|---|---|
| T<br>D<br>F<br>S | Tool number<br>Cutting edge (tool data)<br>Feed/Feed rate<br>Speed/Cutting speed |
| **Path information/departure commands** | |
| **Instruction** | **Meaning** |
| G94 | Linear feed in mm/min * |
| G97 | Spindle speed in $min^{-1}$ * |
| **Switching information** | |
| **Instruction** | **Meaning** |
| M00 | Programmed stop |
| M03<br>M04<br>M05 | Work spindle ON, clockwise<br>Work spindle ON, anti-clockwise<br>Work spindle Stop |
| M06 | Tool change |
| M08<br>M09 | Coolant ON<br>Coolant OFF |
| M17<br>M30 | End of subprogram<br>End of program, jump back to the beginning of program |
| All instructions mentioned above are **modal**, until they are programmed to deactivate with different set of instructions. | |

Notes