

Programming: Q Parameters

9.1 Principle and overview of functions

9.1 Principle and overview of functions

With Q parameters you can program entire families of parts in a single NC program by programming variable Q parameters instead of fixed numerical values.

Use Q parameters for e.g.:

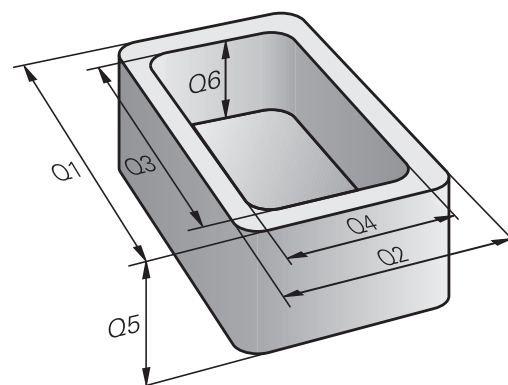
- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

With Q parameters you can also:

- Program contours that are defined through mathematical functions
- Make execution of machining steps depend on certain logical conditions
- Variably design FK programs

Q parameters are always identified with letters and numbers. The letters determine the type of Q parameter and the numbers the Q parameter range.

For more information, see the table below:



Q parameter type	Q parameter range	Meaning
Q parameters:		
	0 - 99	Parameters effect all NC programs in the TNC memory Parameters for the user , if there are no overlaps with the HEIDENHAIN-SL cycles
	100 - 199	Parameters for system information on the TNC that can be read by the NC programs of the user or by cycles
	200 - 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 - 1399	Parameters that are primarily used with manufacturer cycles when values are given back to the user program
	1400 - 1599	Parameters primarily used as input parameters for manufacturer cycles
	1600 - 1999	Parameters for users
QL parameters:		
	0 - 499	Parameters only effective locally within an NC program Parameters for users
QR parameters:		
	0 - 499	Parameters that are nonvolatile on all NC programs in the TNC memory, i.e. they remain in effect even after a power interruption Parameters for users

QS parameters (the **S** stands for string) are also available on the TNC and enable you to process texts.

Q parameter type	Q parameter range	Meaning
QS parameters:		Parameters effect all NC programs in the TNC memory
	0 - 99	Parameters for the user , where no overlaps with the HEIDENHAIN SL cycles are present
	100 - 199	Parameters for system information on the TNC that can be read by the NC programs of the user or by cycles
	200 - 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 - 1399	Parameters that are primarily used with manufacturer cycles when values are given back to the user program
	1400 - 1599	Parameters primarily used as input parameters for manufacturer cycles
	1600 - 1999	Parameters for users



You gain maximum safety for your applications by using only Q parameter ranges recommended for the user in your NC programs.

Please note that the specified use of the Q parameter ranges is recommended by HEIDENHAIN but cannot be ensured.

Machine manufacturer or third-party functions may still cause overlaps with the user's NC program. Please refer to the machine manual and third-party documentation for this.