



# HEIDENHAIN



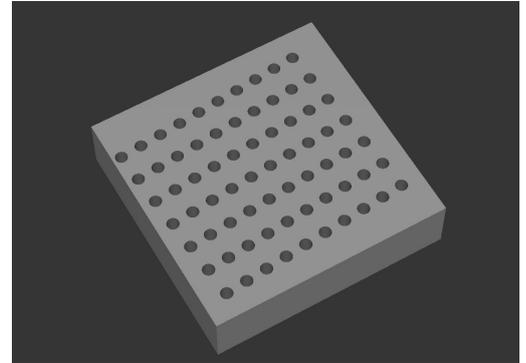
## NC Solutions

Description of NC program 1020

English (en)  
4/2017

## 1 Description of the NC program 1020\_en.h

NC program for defining a point pattern as grid lines.



### Description

With this NC program the control generates a point pattern in the form of grid lines. The control calls a machining cycle at the calculated positions, enabling you to simply select the type of machining.

In the first part of the NC program you define all parameters required for the calculation, the tool, and the machining cycle executed by the control at the calculated positions. The control then calls a subprogram. The control executes all calculations and positioning movements in this subprogram. The control calculates the positions so that it approaches the positions in a meandering path for machining. Define the position of the first machining via the parameters. After the last machining step the control retracts the tool and terminates the program.

Parameter	Name	Meaning
Q1	STARTING POINT IN THE X AXIS	X coordinate at which the control executes the first machining step
Q2	STARTING POINT IN THE Y AXIS	Y coordinate at which the control executes the first machining step
Q3	DISTANCE OF MACHINING STEPS IN X	Incremental clearance of machining in the X axis
Q4	DISTANCE OF MACHINING STEPS IN Y	Incremental clearance of machining in the Y axis
Q5	NUMBER OF MACHINING STEPS IN X	Number of machining steps the control executes in each row in the X axis
Q6	NUMBER OF MACHINING STEPS IN Y	Number of machining steps the control executes in each column in the Y axis
Q7	ROTATION	Rotation of the coordinate system around the first machining position
Q8	SAFETY CLEARANCE	Z clearance between the tool and workpiece surface approached by the control in rapid traverse before machining is executed

