



# HEIDENHAIN



## NC Solutions

Description of NC Program 4040

English (en)  
9/2017

## 1 Description of NC program 4040\_en.h

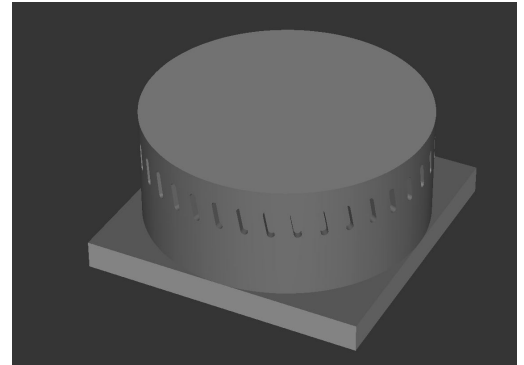
NC program used by the control to repeatedly execute a machining cycle on a cylinder surface.



The NC program is written for an A/C kinematics configuration.



In this NC program, the control repeats a machining cycle on the lateral surface of a cylinder. No compensation movement of the rotary axes takes place while the cycle is being executed. This means that it is not a cylinder surface interpolation.



### Requirement:

Execute Cycle 253 SLOT MILLING repeatedly on a cylinder surface.

### NC program 4040\_en.h:

In this NC program, the control first uses Cycle 257 **CIRCULAR STUD** to machine the workpiece. You define the stud in the cycle. If the workpiece blank is a cylinder, this step may be omitted.

You subsequently define the tool for machining the cylinder surface. The control positions the tool to a safe position. Then the control positions the first rotary axis with **PLANE AXIAL** so that the tool axis is perpendicular to the cylinder surface.

After that, another **PLANE AXIAL** is defined in a program section repeat. This function is used by the control to incrementally move the second rotary axis with every repetition. Then the control uses a CALL LBL to call a subprogram. The machining cycle is defined in this subprogram. The control then pre-positions the tool to the starting point and executes the cycle.

After the number of defined program section repeats has been reached, the control retracts the tool. The control subsequently cancels the tilting of the working plane and ends the NC program.



Please note while programming:

- The preset in the X axis and Y axis must be in the center of the workpiece
- When defining the cycle parameters, please make sure to enter the cylinder radius into the surface coordinate parameter

