HEIDENHAIN
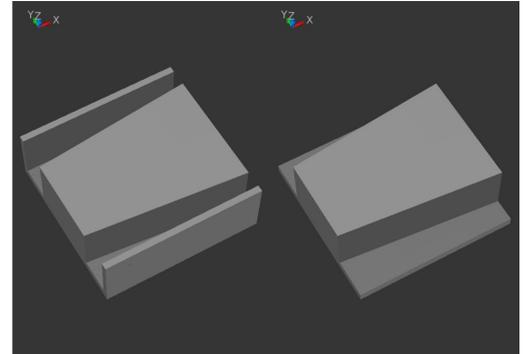
NC solutions

Description of NC Program 2145

1 Description of the NC program 2145_en.h

An NC program in which the contour for an SL cycle can be selected using a Q-Parameter.

Requirement

The SL cycle should be used to mill a stud. A Q-Parameter is to be used to select which contour should be used as a boundary for the pocket, depending on the workpiece blank.

NC program 2145_en.h

In the NC program, you first define the tool. Then the control moves to the safety clearance. You then define a parameter with the number of the subprogram that describes the contour that is to be used.

In the next program section, the SL cycles are defined. In Cycle 14 **CONTOUR**, two contour labels are specified in the example program. The contour for the island is programmed in a subprogram, and the control selects the pocket contour to be used in the second subprogram. This is why, in this contour label, a **CALL LBL** is defined that, with the parameter you have defined, calls the subprogram containing the contour to be used.

You then define the machining parameters in Cycles 20 and 22. The control calls the cycles with **CYCL CALL** and machines the contours. In the final program step, the control retracts the tool and ends the NC program.

The subprograms are defined after the end of the program. In the example program, one subprogram with the contour for the island, two subprograms for the different pocket contours, and one in which **CALL LBL** is defined—with which the contour selection is implemented.

Parameter	Name	Meaning
Q30	SELECTION OF THE POCKET CONTOUR	Number of the subprogram in which the contour to be used is defined.