



HEIDENHAIN



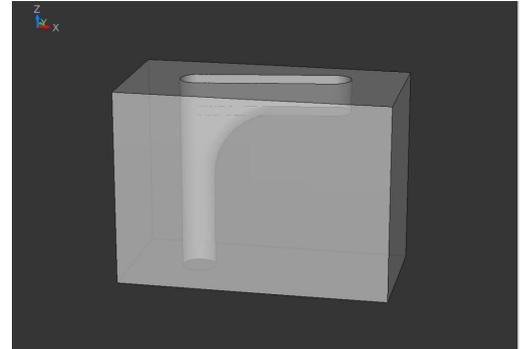
NC solutions

Description of NC Program 3200

English (en)
9/2017

1 Description of the NC program 3200_en.h

NC program for machining a cable feedthrough with kinking protection



Description

With this NC program, the control mills a cable feedthrough with kinking protection. This cable feedthrough consists of a hole, a slot, and a radius rounding the hole and the slot together. The control creates the radius using discrete linear elements. You define the number of elements making up the radius in a parameter. This definition allows you to influence the surface quality. The complete machining operation is carried out with a tool.



Tool notes

- The tool must be suitable for drilling and milling operations.
- The length of the cutting edge of the tool must be larger than the defined rounding radius.

At program start, you define the tool and all of the parameters required for machining. The control then moves the tool to a safe position. Subsequently, the control performs several calculations. In the next step, the control shifts the datum to the center of the hole and rotates the coordinate system by the defined rotation.

After this, the control positions the tool at the center of the hole and then moves it to the safety clearance. In the following step, the control moves the tool to the drilling depth at the feed rate. When the drilling depth has been reached, the control retracts the tool to the depth of the slot. The control then mills the slot. Always ensure that the machining operation is in the positive direction of the X axis in the coordinate system. You define the position of the slot with the ROTATION parameter. At the end of the slot, the control retracts the tool to the safety clearance.

It then repositions the tool at the center of the hole and subsequently moves it along the Z axis to the starting point of the rounding radius. In a program section repeat, the control traverses an incremental linear path. The control repeats the program section until the number you defined has been reached.

When the rounding radius has been completed, the control retracts the tool to the safety clearance. It then moves the tool to a safe height. In the final program section, the tool resets the rotation and the datum shift and ends the NC program.

Parameter	Name	Meaning
Q1	X POSITION OF HOLE	X coordinate of the hole center
Q2	Y POSITION OF HOLE	Y coordinate of the hole center
Q3	ANGLE OF ROTATION	Rotation of the slot, with absolute reference to the X+ direction
Q4	SAFETY CLEARANCE	Z coordinate that the control approaches in rapid traverse
Q5	TOTAL HOLE DEPTH	Depth of the drilling operation
Q6	FEED RATE FOR PLUNGING	Traversing speed of the tool in the Z axis
Q7	SLOT DEPTH	Depth of the milling operation of the slot
Q8	FEED RATE FOR RETRACTION	Traversing speed of the tool during retraction
Q9	SLOT LENGTH	Slot length as measured from the hole center to the circle center at the end of the slot
Q10	FEED RATE FOR MILLING	Traversing speed of the tool in the X/Y plane
Q11	ROUNDING RADIUS	Radius at which the hole and the slot are rounded
Q12	NO. OF STEPS IN THE RADIUS	The number of linear movements into which the control divides the radius

