



Do you know this function?

A reliable process for the milling of fits

Reliably manufacturing fits in series production

Klartext presents a method for the precise and reliable manufacture of fits—particularly in mid-size and large series runs. The challenge is the fact that cutting conditions change continuously. Particularly the cutting pressure changes during milling as the result of increasing tool wear. The values for the tool dimensions must continuously be adapted because of this. Along with the usual tool measurement, the method presented here also takes into account the current cutting conditions, since the actual dimensions of the workpiece are measured. This is done

automatically, without repeatedly needing to manually adapt the tool's compensation values.

The recommendation is to use touch-probe cycles 421 through 430. This is very convenient, since tool monitoring can be activated in these cycles. The control then performs continuous tool compensation automatically. How often should the measurement be repeated? You decide this individually, depending on the machining task.

Details about the method

First the milled fit is measured with a touch probe. It is important that the fit is roughed and pre-finished (same finishing allowance as for actual finishing). Based on the values measured, the control corrects the tool compensation values in the tool table—meaning the oversize DR for the tool radius or DL for the length. The cutting pressure has already been accounted for in this compensation, since the actually machined workpiece was measured.

Now you call the tool again and the fit is completed. The control takes the compensation values previously determined into account.

How is it ensured that this compensation is reliable? The recommendation here is to find an appropriate number of workpieces after which the touch-probe cycle is called again, e.g. after every fifth workpiece. The program section with the touch-probe cycle is simply controlled by a counter, for example by incrementing in QR parameters.

With each new measuring process the tool compensation values are adapted to the current situation.

Reliably producing the first workpiece

This strategy also includes the first fit, so that the first workpiece doesn't end up as scrap. For the first measuring cut you simply enter a greater oversize for the milling tool: Choose such a large value that the next finishing cut encounters similar cutting conditions.

Avoiding tool breakage

By the way, this method also monitors the tool. The cutting pressure continuously increases, theoretically until the tool breaks. Here the control lets you enter maximum delta values. When this value is reached, the control locks the tool and activates a replacement tool, if desired.

+ You can find example programs and other information in our NC database at <http://applications.heidenhain.de/ncdb>

It doesn't get any more exact than this: In the probe cycle you define the maximum and minimum limits as well as the tolerance values for the fit (Q277 through Q280). If tool monitoring is activated (Q330), the TNC corrects the tool radius in the tool table, depending on the deviation from the nominal value.

The screenshot displays the 'Programming and editing' screen of a TNC 530. The title bar reads 'Maximum limit of size for stud?'. The program code is as follows:

```

0385=+1500 ;FINISHING FEED RATE
60 L X+50 Y+33 R0 FMAX M99
69 STOP
70 QR10 = QR10 + 1
71 FN 12: IF +QR10 LT +4 GOTO LBL 99
72 QR10 = 0 ;RESET QR10
73 * - MEASURE
74 TOOL CALL "3D-PROBE" Z
75 TCH PROBE 422 MEAS. CIRCLE OUTSIDE
  Q273=+00 ;CENTER IN 1ST AXIS
  Q274=+00 ;CENTER IN 2ND AXIS
  Q282=+19.994 ;NOMINAL DIAMETER
  Q325=+0 ;STARTING ANGLE
  Q247=+90 ;STEPPING ANGLE
  Q281=-4 ;MEASURING HEIGHT
  Q320=+3 ;SET-UP CLEARANCE
  Q280=+50 ;CLEARANCE HEIGHT
  Q301=+0 ;MOVE TO CLEARANCE
  Q277=20 ;MAXIMUM LIMIT
  Q278=+18.997 ;MINIMUM LIMIT
  Q279=+0 ;TOLERANCE 1ST CENTER
  Q280=+0 ;TOLERANCE 2ND CENTER
  Q281=+2 ;MEASURING LOG
  Q309=+0 ;PGM STOP TOLERANCE
  Q330=+0 ;TOOL
  Q423=+4 ;NO. OF MEAS. POINTS
  Q365=+1 ;TYPE OF TRAVERSE
76 TCH PROBE 421 MEASURE HOLE
  Q279=+35 ;CENTER IN 1ST AXIS
  Q274=+70 ;CENTER IN 2ND AXIS
  Q282=+30.005 ;NOMINAL DIAMETER
  Q325=+0 ;STARTING ANGLE
  Q247=+90 ;STEPPING ANGLE
  Q281=-4 ;MEASURING HEIGHT
  Q320=+3 ;SET-UP CLEARANCE
  Q280=+50 ;CLEARANCE HEIGHT
  Q301=+0 ;MOVE TO CLEARANCE
  Q275=+30.013 ;MAXIMUM LIMIT
  Q276=+30 ;MINIMUM LIMIT
  Q279=+0 ;TOLERANCE 1ST CENTER
  Q280=+0 ;TOLERANCE 2ND CENTER
  Q281=+2 ;MEASURING LOG
  Q309=+0 ;PGM STOP TOLERANCE
  Q330=+8.1 ;TOOL
  Q423=+4 ;NO. OF MEAS. POINTS
  Q365=+1 ;TYPE OF TRAVERSE
77 TCH PROBE 427 MEASURE COORDINATE
    
```

The technical drawing shows a hole with diameter Q277. The tool table shows a correction from 0163 to 0161. The S100% and F100% indicators are OFF.