



HEIDENHAIN

Webinar



Deburr components

TNC Controls

WEBINAR



CONTENT

1. Overview of possible applications
2. Deburring with NC-drill
3. Attaching chamfer to contour
4. Apply radius to contour



1 Application possibilities at a glance



Application possibilities

At a glance

- Deburring contours with NC-drills (90° point angle)
- Cutting position on the tool can be selected as required
- NC program example for deburring contours with any angle
- NC program example for deburring contours with any radius



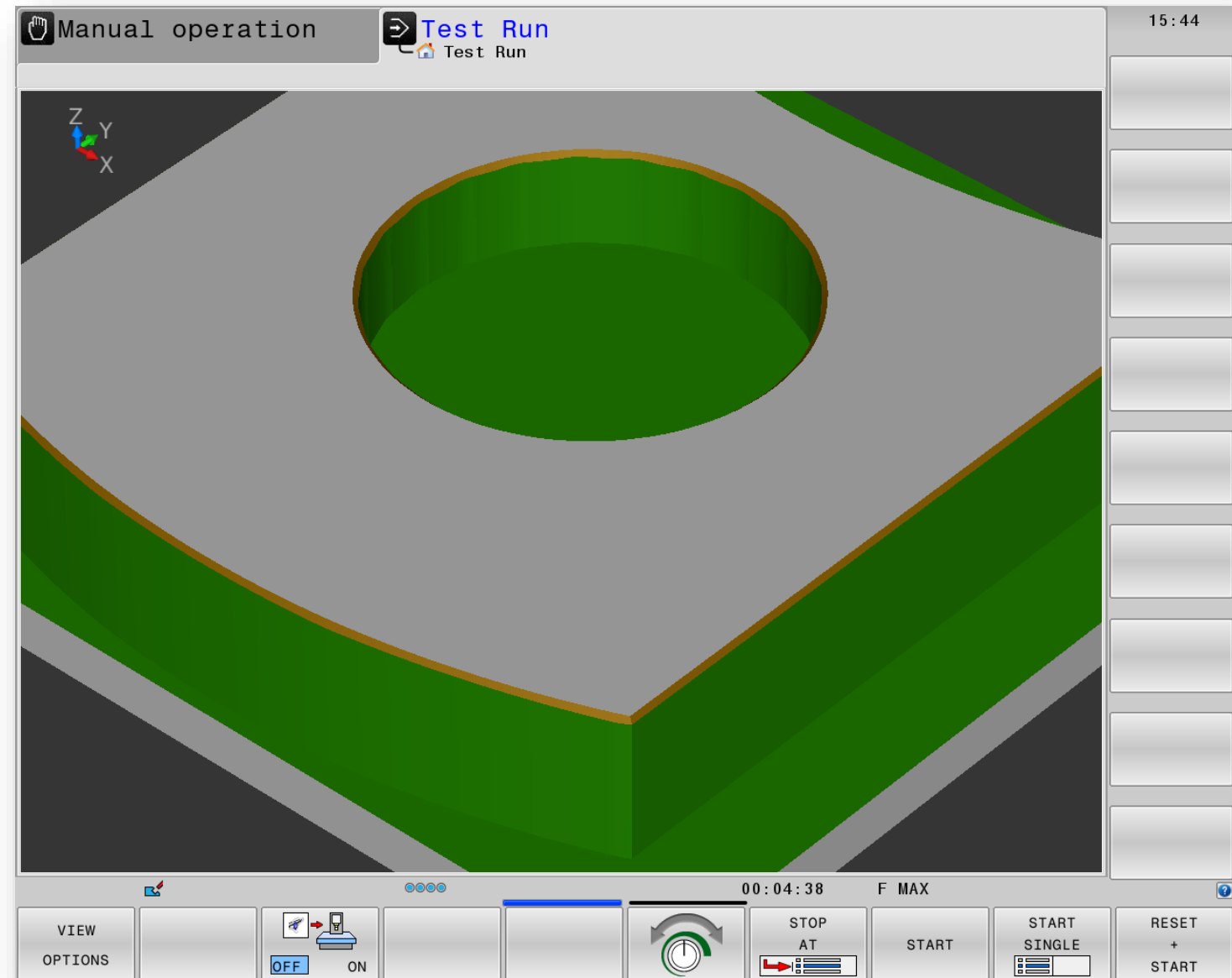


2 Deburring with NC-drill



Deburring With NC-drill

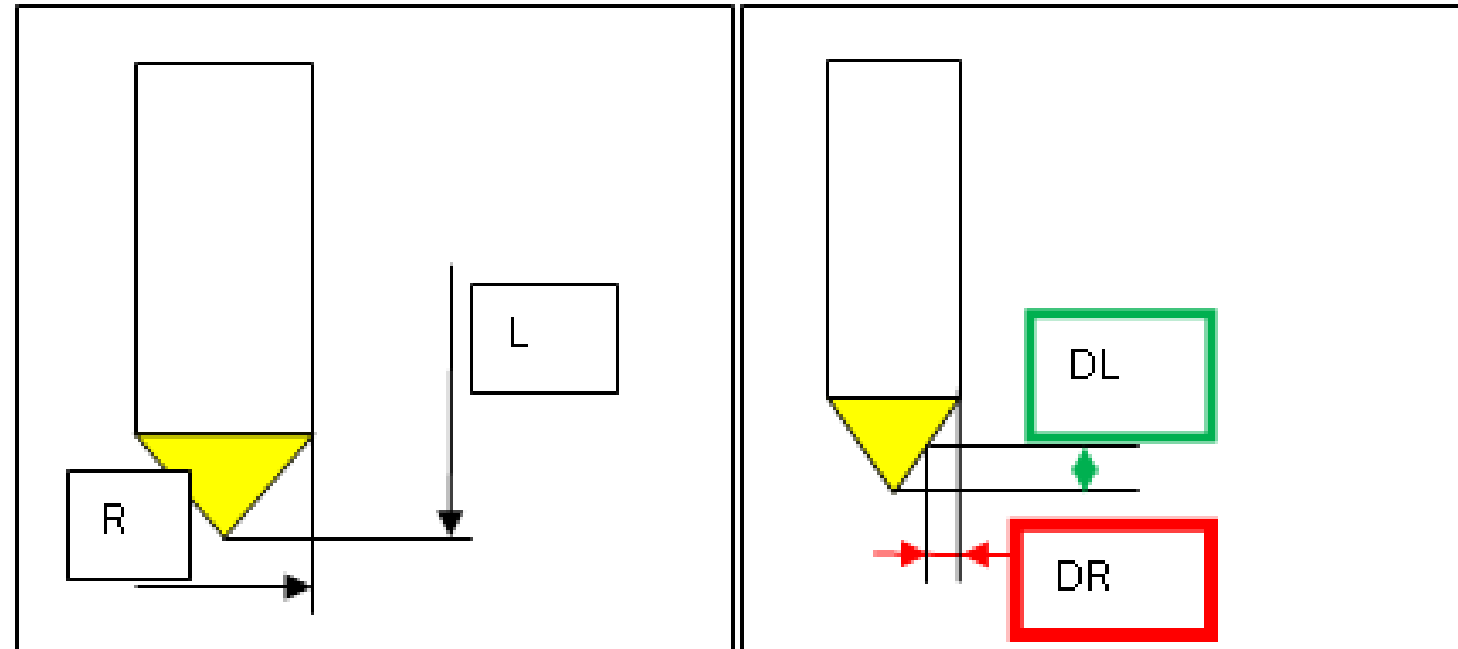
- Measure tool length to theoretical tip
- External tool radius
- Contact point can be varied via DL/DR
- $DL + DR$ must always give the tool radius





Deburring With NC-drill

- Contact point can be varied via DL/DR
- Contours/cycles are finished with radius compensation
- Depth corresponds to the chamfer size





3 Attaching chamfer to contour



Any bevel angle along contours

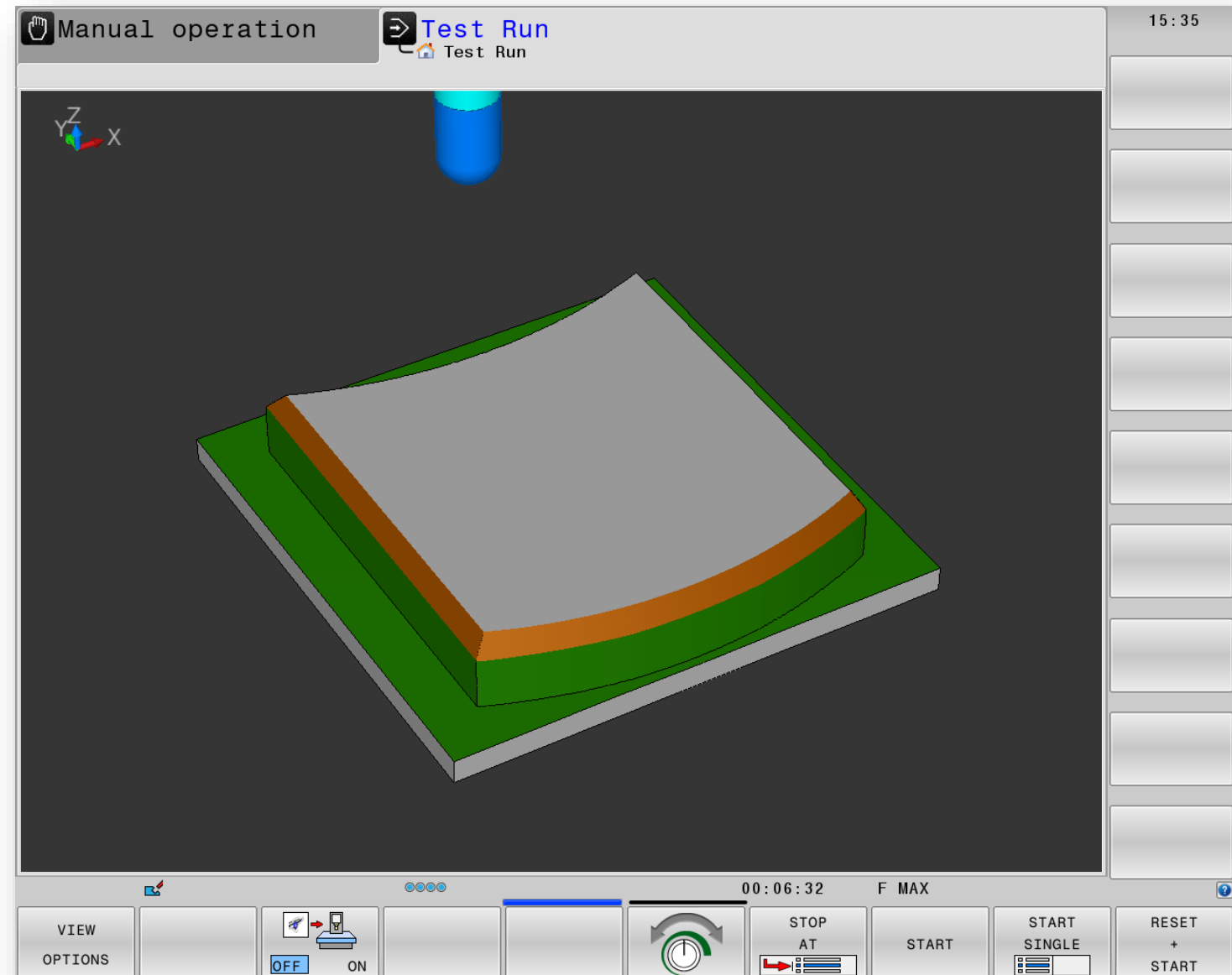
- Finished program example from the NC database
- Fill in charging parameters
- The program calculates all sequences
- The following types of tools can be used:
end mills, ball end mills or torus mills
- Link:
<https://www.klartext-portal.com/en/tips/nc-solutions/#/en/eintraege/fase-an-kontur-fraesen.html>

The screenshot displays the Klartext software interface. At the top, the title bar reads "Klartext" on the left and "HEIDENHAIN" on the right. The main content area features a 3D CAD model of a chamfered part. Below the model, the title "Milling a chamfer on a contour" is followed by the program ID "NC3255". A detailed description explains that the program performs multipass-milling of a chamfer on the upper edge of a contour, first using Cycle 25 and then milling the chamfer in contour lines. It notes that the control reads the cutting edge radius from the tool table to calculate the tool path. Under "Program description", there are three PDF links: "Description_of_NC_program_3255.pdf", "Tool_Table.pdf", and "Notes_for_using_the_NC_Solutions.pdf". The "Program files" section shows a dropdown menu with "3255_en.h" selected. The "Control" section lists "TNC 640", "TNC 620", and "TNC 320" as supported controls. An "OK" button is located in the bottom right corner of the window. The status bar at the very bottom shows "TNC 410".



Any bevel angle along contours

- Finished program example from the NC database
- Fill in charging parameters
- The program calculates all sequences
- The following types of tools can be used:
end mills, ball end mills or torus mills
- Link:
<https://www.klartext-portal.com/en/tips/nc-solutions/#/en/eintraege/fase-an-kontur-fraesen.html>





4 Apply radius to contour



Any radius along contours

- Finished program example from the NC database
- Fill in charging parameters
- The program calculates all sequences
- The following types of tools can be used:
end mills, ball end mills or torus mills
- Link:

<https://www.klartext-portal.com/en/tips/nc-solutions/#/en/eintraege/radius-an-kontur-fraesen.html>

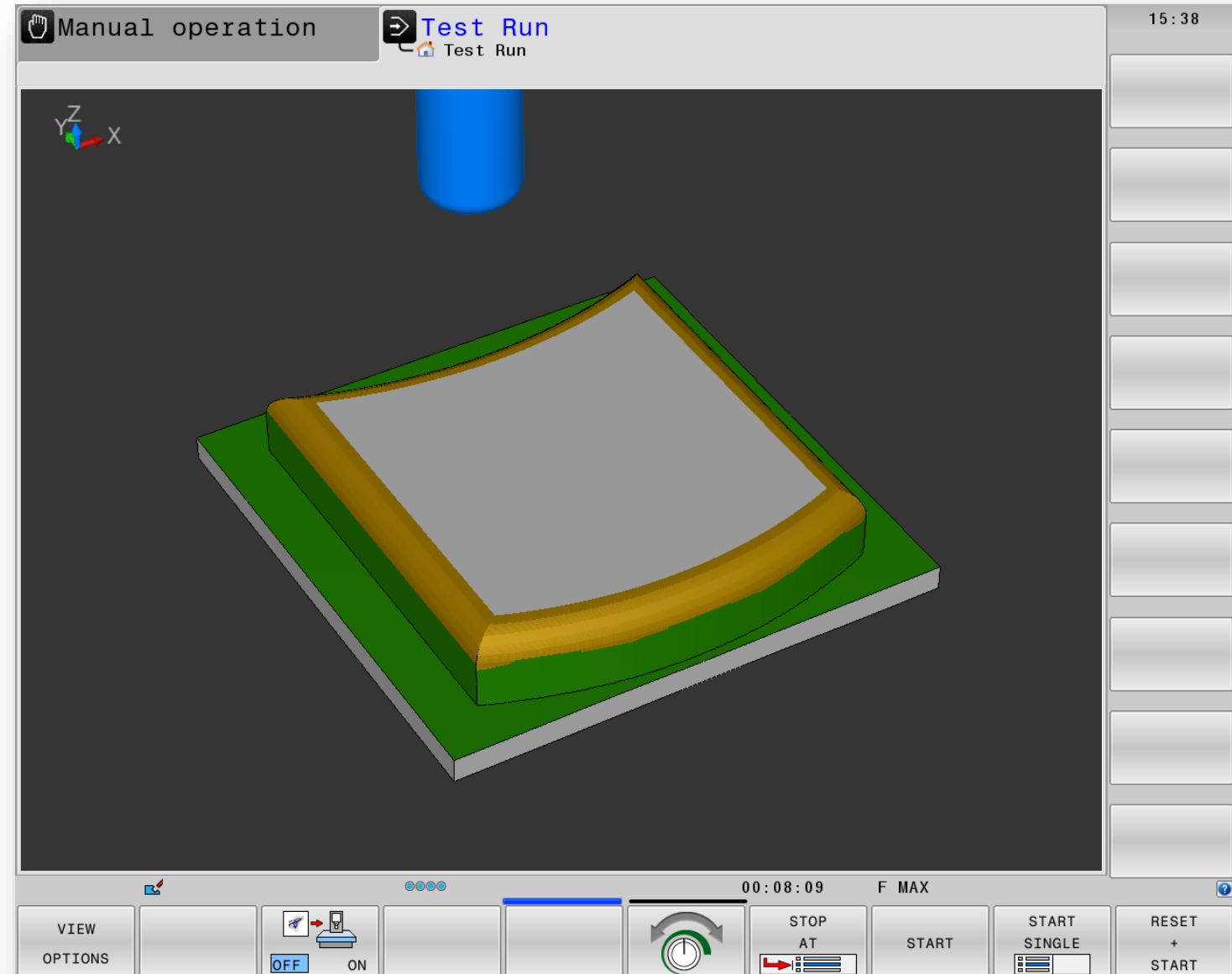
The screenshot shows the Klartext web portal interface. At the top, there is a green header with 'Klartext' on the left and 'HEIDENHAIN' on the right. Below the header, there is a dark grey content area. On the left side of this area, there is a 3D CAD model of a milled part with a radius on its top edge. To the right of the model, the title 'Milling a radius on a contour' is displayed in white. Below the title, the program ID 'NC3250' is shown. A paragraph of text describes the program: 'This NC program is for multipass-milling a radius on the upper edge of a contour. At first, the control machines the contour with Cycle 25. Then it mills the radius in contour lines. The milling of the radius can be performed with an end mill, a ball-nose cutter, or a toroid cutter. For this purpose, the control reads the tool cutter radius from the tool table and calculates the corresponding tool paths, without you having to make any changes to the NC program.' Below this text, there is a section titled 'Program description' with three PDF links: 'Description_of_NC_program_3250.pdf', 'Tool Table.pdf', and 'Notes for using the NC-Solutions.pdf'. Underneath, there is a section titled 'Program files' with a dropdown menu showing '3250_en.h'. At the bottom of the content area, there is a 'Control' section with three buttons: 'TNC 640', 'TNC 620', and 'TNC 320'. In the bottom right corner of the content area, there is an 'OK' button. The footer of the page shows '© TNC 410'.



Any radius along contours

- Finished program example from the NC database
- Fill in charging parameters
- The program calculates all sequences
- The following types of tools can be used:
end mills, ball end mills or torus mills
- Link:

<https://www.klartext-portal.com/en/tips/nc-solutions/#/en/eintraege/radius-an-kontur-fraesen.html>





HEIDENHAIN

Webinar



Contact me:

Phone +49 8669 31-3103

E-Mail 3103@heidenhain.de

WEBINAR