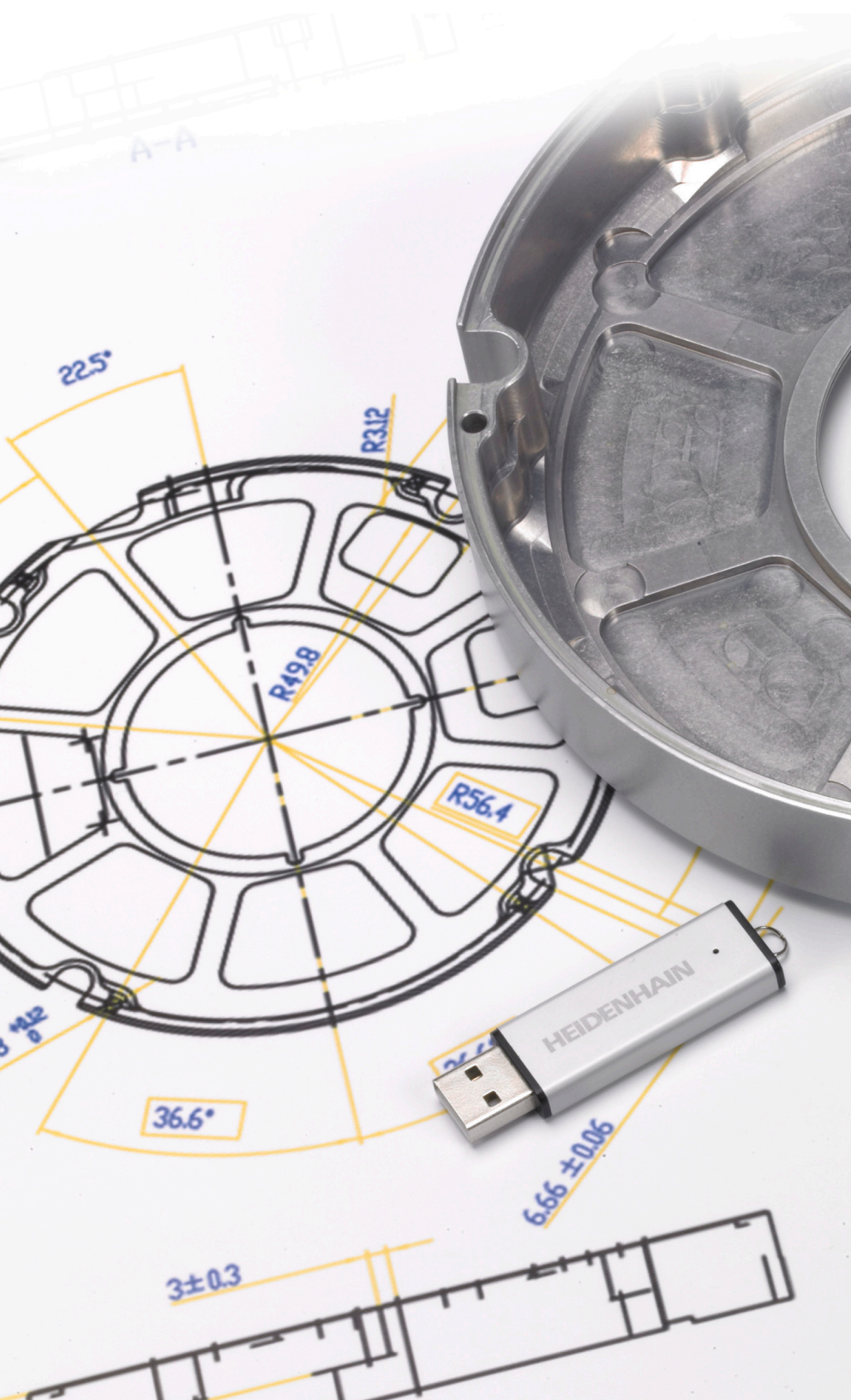




# HEIDENHAIN

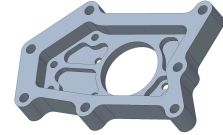


## NC Solutions

Description of NC program 13\_006

## 1.1 Description of NC program 13\_006

NC program for machining an adapter housing. The outside contour, the contours of the pockets, and the hole positions were derived from the 3-D model using the CAD Import option (option 42) and saved in Klartext programs and point tables.



**NC program 13\_006\_en.h**

All of the tools and machining cycles for machining the adapter housing are programmed in the NC program 13\_006\_en.h.

The workpiece is machined in the following steps:

- ▶ Definition of the global settings
  - Global definition 125 POSITIONING
- ▶ Face milling
  - Definition of Cycle 233 FACE MILLING
  - Pre-positioning of the tool and cycle call
- ▶ Roughing the outside contour
  - Definition of the NC program for the outside contour by using the **SEL CONTOUR** function
  - Definition of Cycle 270 CONTOUR TRAIN DATA
  - Definition of Cycle 25 CONTOUR TRAIN
  - Cycle call
- ▶ Roughing the residual material of the outside contour
  - Definition of the NC program for the outside contour by using the **SEL CONTOUR** function
  - Definition of Cycle 270 CONTOUR TRAIN DATA
  - Definition of Cycle 25 CONTOUR TRAIN
  - Cycle call
- ▶ Finishing the outside contour
  - Definition of Cycle 25 CONTOUR TRAIN
  - Cycle call
- ▶ Roughing out the pockets
  - Definition of the NC programs and the depths of the pocket contours by using the **CONTUR DEF** function
  - Definition of Cycle 20 CONTOUR DATA
  - Definition of Cycle 22 ROUGH-OUT
  - Cycle call
- ▶ Finishing the pockets
  - Definition of the NC program for the first pocket contour by using the **SEL CONTOUR** function
  - Definition of Cycle 270 CONTOUR TRAIN DATA

- Definition of Cycle 25 CONTOUR TRAIN
- Cycle call
- Definition of the NC program for the second pocket contour by using the **SEL CONTOUR** function
- Definition of the depth of the second pocket in parameter Q1
- Cycle call
- Definition of the NC program for the third pocket contour by using the **SEL CONTOUR** function
- Definition of the depth of the third pocket in parameter Q1
- Cycle call
- Definition of the NC program for the fourth pocket contour by using the **SEL CONTOUR** function
- Definition of the depth of the fourth pocket in parameter Q1
- Cycle call
- Definition of the NC program for the fifth pocket contour by using the **SEL CONTOUR** function
- Definition of the depth of the fifth pocket in parameter Q1
- Cycle call
- Definition of the NC program for the sixth pocket contour by using the **SEL CONTOUR** function
- Definition of the depth of the sixth pocket in parameter Q1
- Cycle call
- ▶ Hole centering
  - Definition of the point table for centering by using the **SEL PATTERN** function
  - Definition of Cycle 240 CENTERING
  - Cycle call at all positions defined in the table
- ▶ Drilling of holes (diameter: 4 mm)
  - Definition of the point table for the holes (diameter: 4 mm) by using the **SEL PATTERN** function
  - Definition of Cycle 203 UNIVERSAL DRILLING
  - Cycle call at all positions defined in the table
- ▶ Drilling of holes (diameter: 6 mm)
  - Definition of the point table for the holes (diameter: 6 mm) by using the **SEL PATTERN** function
  - Definition of Cycle 203 UNIVERSAL DRILLING
  - Cycle call at all positions defined in the table
- ▶ Deburring
  - Definition of the point table for the holes (diameter: 6 mm) by using the **SEL PATTERN** function
  - Definition of Cycle 252 CIRCULAR POCKET

- Cycle call at all positions defined in the table
- Definition of the NC program for the outside contour by using the **SEL CONTOUR** function
- Definition of Cycle 270 CONTOUR TRAIN DATA
- Definition of Cycle 25 CONTOUR TRAIN
- Cycle call
- Definition of the NC program for the sixth pocket by using the **SEL CONTOUR** function
- Cycle call
- Definition of the NC program for the fifth pocket by using the **SEL CONTOUR** function
- Definition of the depth of the fifth pocket and the other pockets in parameter Q5
- Cycle call
- Definition of the NC program for the fourth pocket by using the **SEL CONTOUR** function
- Cycle call
- Definition of the NC program for the third pocket by using the **SEL CONTOUR** function
- Cycle call
- Definition of the NC program for the second pocket by using the **SEL CONTOUR** function
- Cycle call
- Definition of the NC program for the first pocket by using the **SEL CONTOUR** function
- Cycle call

### Contour programs

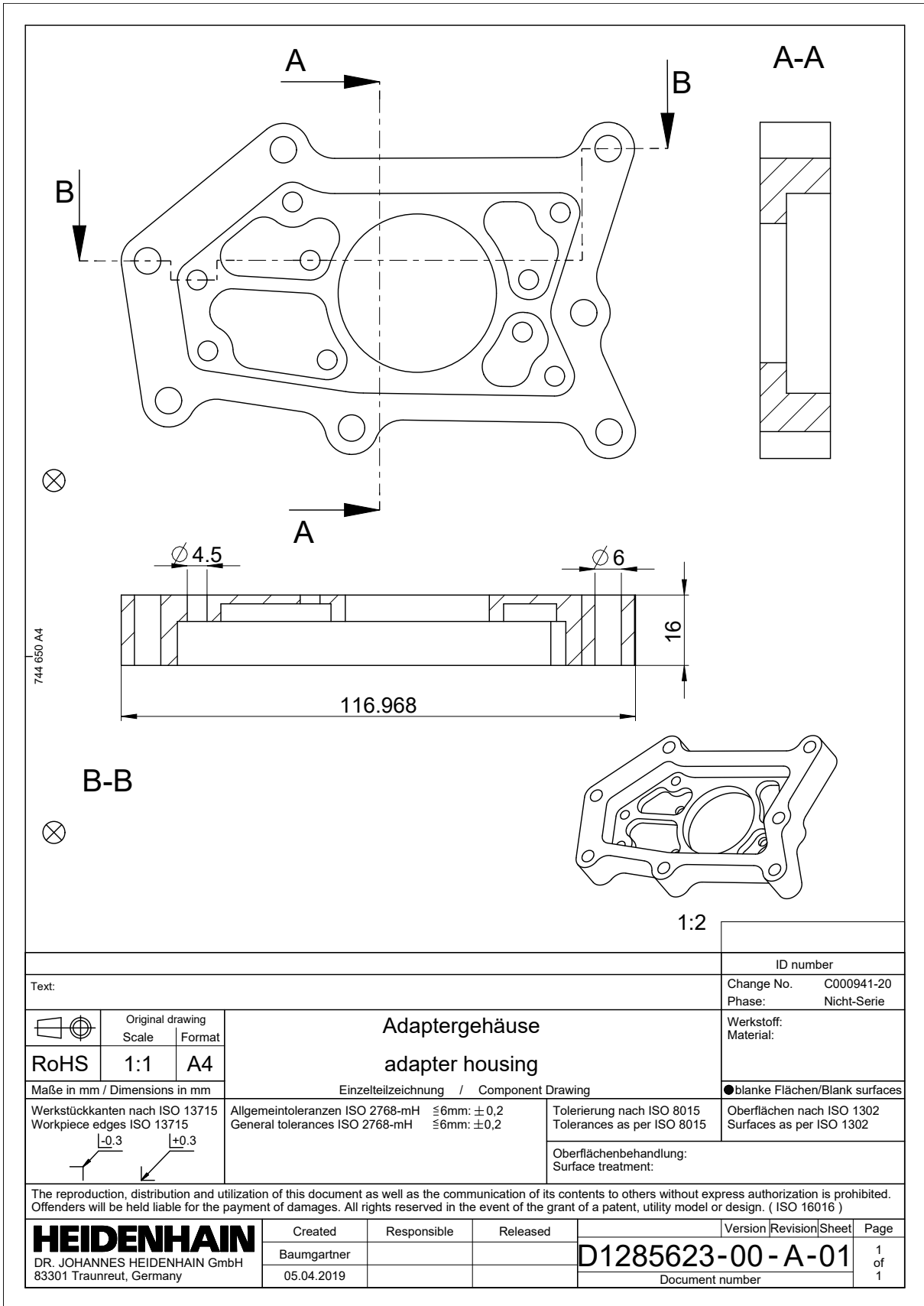
The CAD Import option (option 42) was used to derive the following NC programs from the 3-D model of the adapter housing:

- 13\_006\_Outsidecontour.h
- 13\_006\_Pocket1.h
- 13\_006\_Pocket2.h
- 13\_006\_Pocket3.h
- 13\_006\_Pocket4.h
- 13\_006\_Pocket5.h
- 13\_006\_Pocket6.h

### Point tables

The CAD Import option (option 42) was used to derive the following point tables from the 3-D model of the adapter housing:

- 13\_006\_Centering.PNT
- 13\_006\_Drill\_D4.PNT
- 13\_006\_Drill\_D6.PNT



# 2

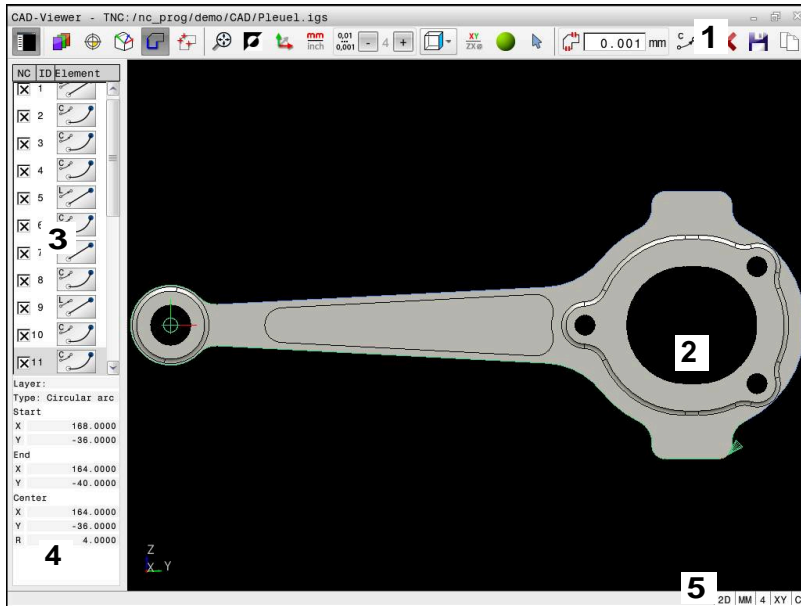
**Excerpt from the  
User's Manual:**

## 2.1 Screen layout of the CAD viewer

### Fundamentals of the CAD viewer

#### Screen display

When you open the **CAD-Viewer**, the following screen layout is displayed:



- 1 Menu bar
- 2 Graphics window
- 3 List View window
- 4 Element Information window
- 5 Status bar

#### File types

The **CAD-Viewer** enables you to open standardized CAD data formats directly on the control.

The control displays the following file types:

File	Type	Format
Step	.STP and .STEP	<ul style="list-style-type: none"> <li>■ AP 203</li> <li>■ AP 214</li> </ul>
IGES	.IGS and .IGES	<ul style="list-style-type: none"> <li>■ Version 5.3</li> </ul>
DXF	.DXF	<ul style="list-style-type: none"> <li>■ R10 to 2015</li> </ul>



## 2.2 CAD Import (option 42)

### Application

You can open CAD files directly on the control in order to extract contours and machining positions from it. You can then store them as Klartext programs or as point files. Klartext programs acquired in this manner can also be run on older HEIDENHAIN controls, since these contour programs contain only **L** and **CC/C** blocks.

**i** As an alternative to **CC** or **C** blocks, you can configure circular movements to be output as **CR** blocks.  
**Further information:** "Basic settings", Page 11

If you process files in **Programming** mode, then the control generates contour programs with the file extension **.H** and point files with the extension **.PNT** by default. You can select the file type in the save dialog.

To insert a selected contour or a selected machining position directly into an NC program, use the control's clipboard. Using the clipboard, you can even transfer the contents to the additional tools (e.g., **Leafpad** or **Gnumeric**).

**i** Operating notes:

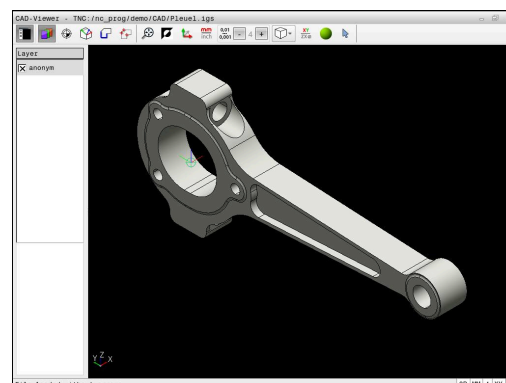
- Before loading the file into the control, ensure that the name of the file contains only permitted characters. **Further information:** "File names", Page
- The control does not support binary DXF format. Save the DXF file in ASCII format in the CAD or drawing program.

### Using the CAD viewer

**i** To use the **CAD-Viewer** without a touchscreen, you have to use a mouse or touchpad.

The **CAD-Viewer** runs as a separate application on the third desktop of the control. This enables you to use the screen switchover key to switch between the machine operating modes, the programming modes, and the **CAD-Viewer**. This is particularly useful if you want to add contours or machining positions to a Klartext program using the clipboard.

**i** If you are using a TNC 640 with touch control, you can replace some keystrokes with gestures.  
**Further information:** "Operating the touchscreen", Page



## Opening the CAD file



- ▶ Press the **Programming** key



- ▶ Press the **PGM MGT** key
- > The control opens the file manager.



- ▶ Press the **SELECT TYPE** soft key
- > The control displays the selectable file formats.



- ▶ Press the **SHOW CAD** soft key
- ▶ Alternative: Press the **SHOW ALL** soft key



- ▶ Select the directory in which the CAD file is saved


















- ▶ Select the desired CAD file

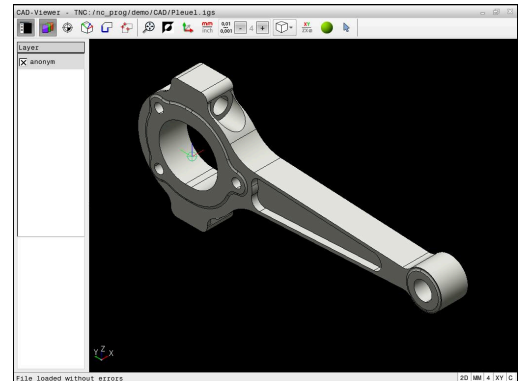





- ▶ Press the **ENT** key
- > The control starts the **CAD-Viewer** and shows the file contents on the screen. The control displays the layers in the List View window and the drawing in the Graphics window.

### Basic settings



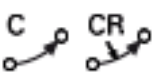



The basic settings specified below are selected using the icons in the header bar.

Icon	Setting
	Show or hide the List View window in order to expand the Graphics window
	Display of the various layers
	Set preset, with optional selection of the plane
	Set datum, with optional selection of the plane
	Select contour
	Select drilling positions
	Set the zoom to the largest possible rendering of the entire graphical representation
	Switch background color (black or white)
	Switch between 2-D and 3-D mode. The active mode is highlighted in color
	Set the unit of measure ( <b>mm</b> or <b>inch</b> ) for the file. The control then outputs the contour program and the machining positions in this unit of measure. The active unit of measure is highlighted in red
	Select the resolution. The resolution defines the number of decimal places and the number of positions for linearization. Default setting: 4 decimal places with <b>mm</b> , and 5 decimal places with <b>inch</b> as the unit of measure
<p> The <b>CAD-Viewer</b> linearizes all of the contours that are not in the XY plane. The finer the resolution, the more accurately the control will display the contours.</p>	
	Switch between various views of the model e.g. <b>Top</b>
	Select a contour for a turning operation. The active machining operation is highlighted in color (option 50)
	Activate wire model of a 3-D drawing



Icon	Setting
  	<p>"Select, add, or remove contour elements" mode</p> <div style="border: 1px solid black; padding: 5px;"> <p><b>i</b> The icon shows the current mode. Clicking the icon activates the next mode.</p> </div>

The control displays the following icons only in certain modes.

Icon	Setting
	The most recent step is undone.
	<p>Contour transfer mode:</p> <p>The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred when the drawing was made. The default setting is 0.001 mm</p>
	<p>Arc mode:</p> <p>Arc mode defines whether circular arcs are output in C format or CR format (e.g., for cylinder surface interpolation) in the NC program.</p>
	<p>Point transfer mode:</p> <p>Specifies whether the control should display the tool path as a dashed line during the selection of machining positions</p>
	<p>Path optimization mode:</p> <p>The control optimizes the tool traverse movement so that there are shorter traverse movements between the machining positions. When the icon is pressed again, the optimization is reset</p>
	<p>Hole position mode:</p> <p>The control opens a pop-up window in which you can filter holes (full circles) based on their size</p>

**i** Operating notes:

- Set the correct unit of measure, because the CAD file does not contain this information.
- When creating NC programs for earlier control models, you must limit the resolution to three decimal places. In addition, you must remove the comments that the **CAD-Viewer** outputs into the contour program.
- The control displays the active basic settings in the status bar of the screen.

## Setting layers

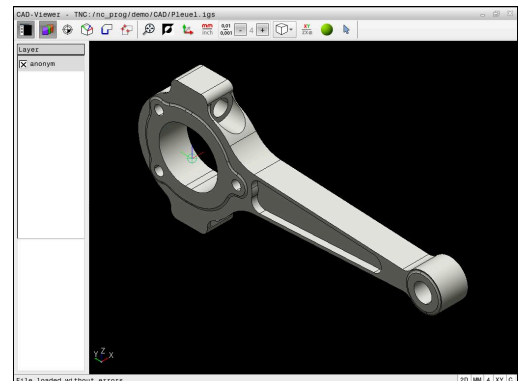
CAD files usually contain several layers. The designer uses these layers to create groups of various types of elements, e.g. the actual workpiece contour, dimensions, auxiliary and design lines, shadings, and texts.

Hiding unneeded layers makes the graphics easier to read and facilitates the extraction of the required information.



### Operating notes:

- The CAD file to be processed must contain at least one layer. Elements not assigned to a layer are automatically moved by the control to the anonymous layer.
- You can even select a contour if the designer has saved the lines on different layers.
- If you double-click a layer, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of a contour.



When you open a CAD file in the **CAD-Viewer**, all layers available are shown.

## Hiding a layer

To hide a layer, proceed as follows:



- ▶ Select the **SET LAYER** function
- > In the List View window the control shows all layers contained in the active CAD file.
- ▶ Select the desired layer
- ▶ Select the check box to deactivate it
- ▶ Alternatively, use the space key
- > The control hides the selected layer.

## Showing a layer

To show a layer, proceed as follows:



- ▶ Select the **SET LAYER** function
- > In the List View window the control shows all layers contained in the active CAD file.
- ▶ Select the desired layer
- ▶ Select the check box to activate it
- ▶ Alternatively, use the space key
- > The control marks the selected layer in the List View with an x.
- > The selected layer is shown.

## Setting a preset

The datum of the drawing in the CAD file is not always located in a manner that lets you use it as a workpiece preset. Therefore, the control provides a function with which you can shift the workpiece preset to a suitable location by clicking an element. You can also define the orientation of the coordinate system.

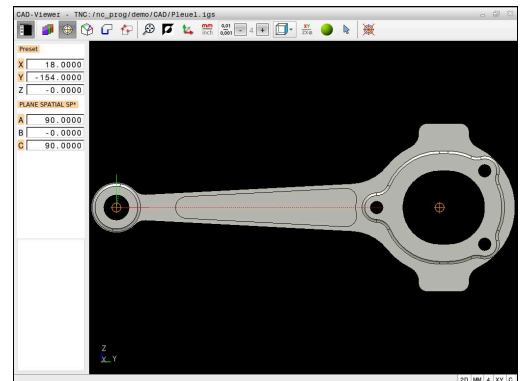
You can position the preset at the following locations:

- By directly inputting numerical values into the List View window
- For straight lines:
  - Starting point
  - Center
  - End point
- For circular arcs:
  - Starting point
  - Center
  - End point
- For full circles:
  - At the quadrant transitions
  - At the center
- At the intersection between:
  - Two straight lines, even if the point of intersection is actually on the extension of one of the lines
  - Straight line and circular arc
  - Straight line and full circle
  - Two circles (regardless of whether a circular arc or a full circle)



### Operating note:

You can change the preset even after you have selected the contour. The control does not calculate the actual contour data until you save the selected contour in a contour program.



## NC syntax

The preset and optional orientation are inserted in the NC program as a comment starting with **origin**.

```
4 ;origin = X... Y... Z...
```

```
5 ;origin_plane_spatial = SPA... SPB... SPC...
```

### Setting a preset on a single element

To set a preset on a single element, proceed as follows:



- ▶ Select the mode for setting a preset
- ▶ Place the mouse pointer on the desired element
- > The control indicates possible locations for the preset on the selected element with stars.
- ▶ Select the star symbol that matches the desired preset position
- ▶ If necessary, use the zoom function
- > The control sets the preset symbol at the selected location.
- ▶ In addition, align the coordinate system as needed

**Further information:** "Adjusting the orientation of the coordinate system", Page 16

### Setting a preset at the intersection between two elements

To set a preset at the intersection between two elements, proceed as follows:




- ▶ Select the mode for setting a preset
- ▶ Select the first element (straight line, full circle, or circular arc) using the left mouse button
- > The control highlights the element.
- ▶ Select the second element (straight line, circle, or circular arc) using the left mouse button
- > The control sets the preset symbol on the intersection.
- ▶ In addition, align the coordinate system as needed

**Further information:** "Adjusting the orientation of the coordinate system", Page 16



Operating notes:

- If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
- If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
- If the control cannot calculate an intersection, it deselects the previously selected element.

Once a preset has been set, the control displays the preset icon with a yellow quadrant .

Use the following icon to delete a preset that has been set .

**Adjusting the orientation of the coordinate system**

The following conditions must be met in order to align the coordinate system:

- Preset has been defined
- There are elements next to the preset that can be used for the desired alignment

The position of the coordinate system is defined by the orientation of the axes.

To align the coordinate system, proceed as follows:



- ▶ Select an element located in the positive X direction using the left mouse button
- > The control aligns the X axis.
- > The control changes the angle in C.
- ▶ Select an element located in the positive Y direction using the left mouse button
- > The control aligns the Y and Z axes.
- > The control changes the angles in A and C.

**i** For angles not equal to 0, the control shows the List View in orange.

**Element information**

The control displays information about the element on the left of the window:

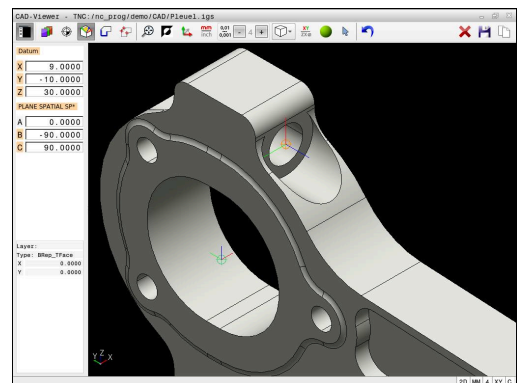
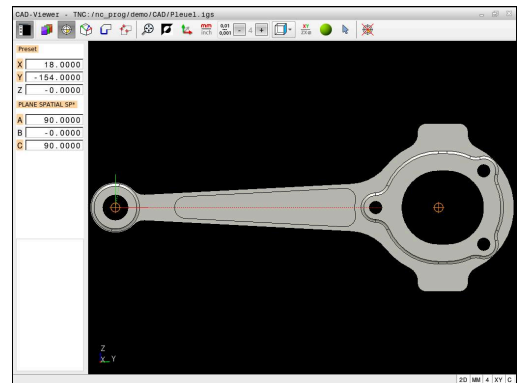
- Distance between the defined preset and the drawing datum
- Orientation of the coordinate system with respect to the drawing

**Setting the datum**

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control has a function with which you can define a new datum and a tilting operation.

The datum with the orientation of the coordinate system can be set at the same positions as a preset.

**Further information:** "Setting a preset", Page 14





**NC syntax**

The datum and its optional orientation can be inserted as NC block or comments in the NC program by using the **TRANS DATUM AXIS** function for the datum and the **PLANE SPATIAL** function for the orientation.

If you define only one datum and its orientation, then the control inserts the functions in the NC program as an NC block.

```
4 TRANS DATUM AXIS X... Y... Z...
```

```
5 PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

If you additionally select contours or points, then the control inserts the functions in the NC program as comments.

```
4 ;TRANS DATUM AXIS X... Y... Z...
```

```
5 ;PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

**Setting the datum on a single element**

To set the datum on a single element, proceed as follows:



- ▶ Select the mode for specifying the datum
- ▶ Place the mouse pointer on the desired element
- > The control indicates possible locations for the datum on the selected element with stars.
- ▶ Select the star symbol that matches the desired datum position
- ▶ If necessary, use the zoom function
- > The control sets the datum icon at the selected location.
- ▶ In addition, align the coordinate system as needed

**Further information:** "Adjusting the orientation of the coordinate system", Page 18

### Setting a datum at the intersection between two elements

To set a datum at the intersection between two elements, proceed as follows:




- ▶ Select the mode for specifying the datum
- ▶ Select the first element (straight line, full circle, or circular arc) using the left mouse button
- > The control highlights the element.
- ▶ Select the second element (straight line, circle, or circular arc) using the left mouse button
- > The control sets the datum icon on the point of intersection.
- ▶ In addition, align the coordinate system as needed

**Further information:** "Adjusting the orientation of the coordinate system", Page 18



Operating notes:

- If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
- If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
- If the control cannot calculate an intersection, it deselects the previously selected element.

Once a datum has been set, the control displays the datum icon with a yellow surface .

Use the following icon to delete a datum that has been set .

### Adjusting the orientation of the coordinate system

The following conditions must be met in order to align the coordinate system:

- The datum has been set
- There are elements next to the preset that can be used for the desired alignment

The position of the coordinate system is defined by the orientation of the axes.

To align the coordinate system, proceed as follows:



- ▶ Select an element located in the positive X direction using the left mouse button
- > The control aligns the X axis.
- > The control changes the angle in C.
- ▶ Select an element located in the positive Y direction using the left mouse button
- > The control aligns the Y and Z axes.
- > The control changes the angles in A and C.



For angles not equal to 0, the control shows the List View in orange.

### Element information

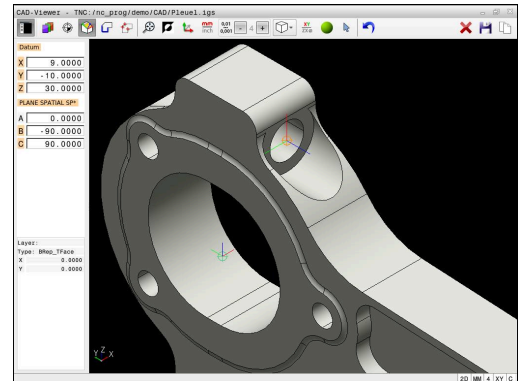
In the Element Information window, the control displays how far away the datum you selected is from the workpiece preset.

The control displays information about the element on the left of the window:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the coordinate system



You can further shift the datum manually after it has been set. To do so, enter the desired axis values into the coordinate field.

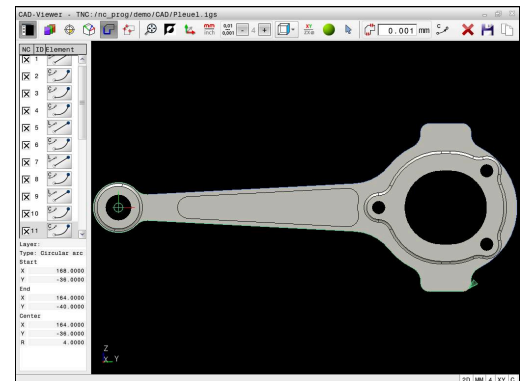


## Selecting and saving a contour



### Operating notes:

- This function is not available if option 42 is not enabled.
- Specify the direction of rotation during contour selection so that it matches the desired machining direction.
- Select the first contour element such that approach without collision is possible.
- If the contour elements are very close to one another, use the zoom function.



The following elements can be selected as a contour:

- Line segment
- Circle
- Circular arc
- Polyline
- Any curves (e.g., splines, ellipses)

### Element information

In the Element Information window the control displays a range of information about the last contour element you selected in the List View window or in the Graphics window.

- **Layer:** specifies the active plane
- **Type:** specifies the type of element (e.g., line)
- **Coordinates:** specify the starting point and end point of an element, and the circle center and radius where appropriate



Ensure that the unit of measure used in the NC program matches with that used in the **CAD-Viewer**. Elements that have been copied from the **CAD-Viewer** to the clipboard do not contain any information about the unit of measure.

## Select contour



### Operating note:

If you double-click a layer in the list view window, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of a contour.

To select a contour using available contour elements, proceed as follows:



- ▶ Select the contour selection mode
- ▶ Place the mouse pointer on the desired element
- > The control displays the suggested direction of rotation as a dashed line.
- ▶ If you need to change the direction of rotation, move the mouse pointer towards the opposite end point
- ▶ Select the element using the left mouse button
- > The selected contour element turns blue.
- > The control shows the other selectable elements in green.



For branched contours, the control chooses the path with the smallest directional deviation. The control provides an additional mode that allows you to modify the suggested contour path.

**Further information:** "Creating contour paths independent of available contour elements", Page 23

- ▶ Select the last green element of the desired contour using the left mouse button
- > The control changes the color of all selected elements to blue.
- > In the List View, all selected elements are given a check mark in the column **NC**.

### Saving a contour



Operating notes:

- The control also transfers two workpiece-blank definitions (**BLK FORM**) to the contour program. The first definition contains the dimensions of the entire CAD file. The second one, which is the active one, contains only the selected contour elements, so that an optimized size of the workpiece blank results.
- The control only saves elements that have been selected (blue elements), which means that they have been given a check mark in the List View window.

To save a selected contour, proceed as follows:



- ▶ Select the Save icon
- ▶ The control prompts you to select the target directory, a file name, and the file type.



- ▶ Enter this information
- ▶ Confirm your input
- ▶ The control saves the contour program.



- ▶ Alternative: Copy the selected contour elements to the clipboard



Ensure that the unit of measure used in the NC program matches with that used in the **CAD-Viewer**. Elements that have been copied from the **CAD-Viewer** to the clipboard do not contain any information about the unit of measure.

### Deselecting the contour

To deselect the selected contour elements, proceed as follows:



- ▶ Select the Clear function to deselect all elements
- ▶ Alternative: Select individual elements by clicking them with the left mouse button while holding the **CTRL** key


### Creating contour paths independent of available contour elements

To select any contours by using the end point, center, or transition points, proceed as follows:



- ▶ Select the contour selection mode



- ▶ Activate the "Add contour elements" mode
- ▶ The control displays the following icon:  

- ▶ Place the mouse pointer on the contour element
- ▶ The control displays selectable points.




#### Selectable points:

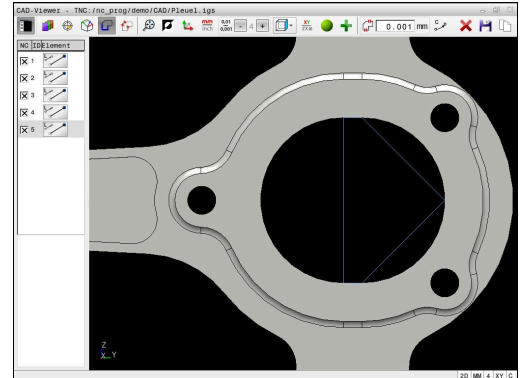
- End point or midpoint of a line or curve
- Quadrant transitions or center of a circle
- Points of intersection between existing elements

- ▶ Select the starting point as needed
- ▶ Select the starting element
- ▶ Select the subsequent element
- ▶ Alternative: Select any selectable point
- ▶ The control creates the desired contour path.



#### Operating notes:

- The contour paths available depend on the selectable contour elements that are shown in green. Without the green elements, the control will display all solutions available. To remove the proposed contour path, select the first green element by pressing the left mouse button while holding the **CTRL** key down. As an alternative, you can switch to the Remove mode:  

- If the contour element to be extended or shortened is a straight line, the control will extend or shorten the contour element along the same line. If the contour element to be extended or shortened is a circular arc, the control will extend or shorten the contour element along the same arc.



### Selecting a contour for a turning operation

You can also use the CAD viewer (option 50) to select contours for turning. The icon is grayed out if option 50 is not enabled. Before selecting a turning contour, you must set the preset on the rotary axis. If you select a turning contour, it is saved with Z and X coordinates. In addition, all X coordinate values in turning contours are transferred as diameter values, i.e. the drawing dimensions for the X axis are doubled. All contour elements below the rotary axis cannot be selected and are highlighted gray.

To select a turning contour using available contour elements, proceed as follows:



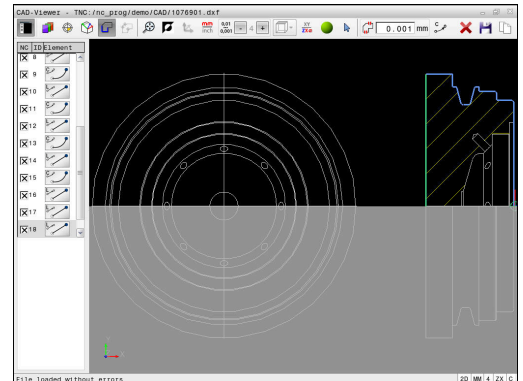
- ▶ Select the mode for choosing a turning contour
- > The control shows only the selectable elements above the rotation center.
- ▶ Select the contour elements using the left mouse button
- > The control displays the selected contour elements in blue.
- > The control also displays the selected elements in the List View window.



Functions or icons that are not available for turning contours appear dimmed.

You can also use the mouse to change the turning graphic display. The following functions are available:

- To shift the displayed model, hold down the center mouse button or the mouse wheel (depending on your mouse model), and move the mouse.
- To zoom in on a certain area mark a zoom area by holding the left mouse button down
- To rapidly zoom in or out rotate the mouse wheel backwards or forwards
- To restore the standard view, double-click with the right mouse button





## Selecting and saving machining positions



### Operating notes:

- This function is not available if option 42 is not enabled.
- If the contour elements are very close to one another, use the zoom function.
- If required, configure the basic settings so that the control shows the tool paths. **Further information:** "Basic settings", Page 11

Three possibilities are available in the pattern generator for defining machining positions:

- Single selection: You select the desired machining positions by clicking them individually with the mouse  
**Further information:** "Single selection", Page 26
- Multiple selection by drag box: You select multiple machining positions by dragging a box around them with the mouse  
**Further information:** "Multiple selection by drag box", Page 26
- Multiple selection by search filter: You select all machining positions within a definable diameter range  
**Further information:** "Multiple selection by search filter", Page 27



Machining positions are deselected, deleted, or saved in the same manner as contour elements.

## Selecting the file type

The following file types are available:

- Point table (.PNT)
- Klartext conversational language program (.H)

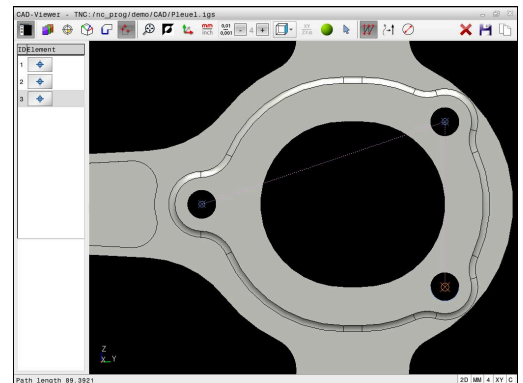
If you save the machining positions to a Klartext program, the control creates a separate linear block with a cycle call for every machining position (**L X... Y... Z... F MAX M99**).



The NC syntax used allows you to export NC programs generated by CAD import to old HEIDENHAIN controls and run them there.



The point tables (.PNT) of the TNC 640 and iTNC 530 are not compatible. Transferring a point table to and running it on the other control model leads to problems and unpredictable behavior.

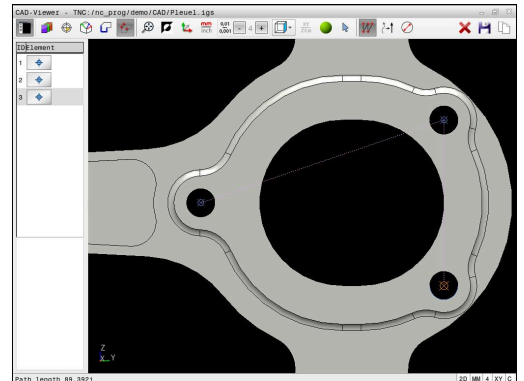


**Single selection**

To select individual machining positions, proceed as follows:




- ▶ Select the mode for choosing a machining position
- ▶ Place the mouse pointer on the desired element
- ▶ The control displays the selectable element in orange.
- ▶ Select the circle center as machining position
- ▶ Alternative: Select the circle or a circle segment
- ▶ The control loads the selected machining position into the List View window.

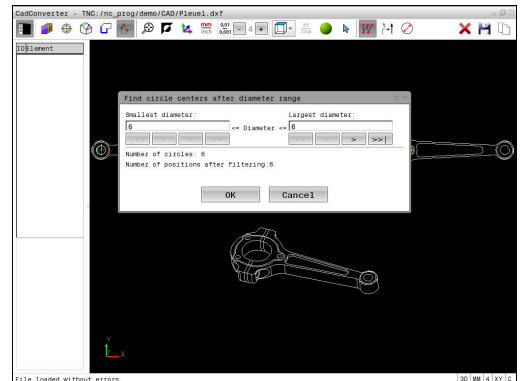


**Multiple selection by drag box**

To select multiple machining positions by dragging a box around them, proceed as follows:




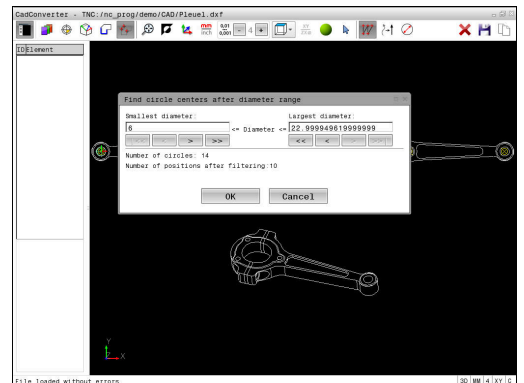
- ▶ Select the mode for choosing a machining position
- ▶ Activate the Add function
- ▶ The control displays the following icon: 
- ▶ Drag a box around the desired area while holding down the left mouse button
- ▶ The control displays the minimum and maximum diameter in a pop-up window.
- ▶ Change the filter settings as needed  
**Further information:** "Filter settings", Page 27
- ▶ Confirm the diameter range with **OK**
- ▶ The control loads all machining position within the selected diameter range into the List View window.



### Multiple selection by search filter

To select multiple machining positions by search filter, proceed as follows:





- 
  - ▶ Select the mode for choosing a machining position
  - ▶ Activate the search filter
  - ▶ The control displays the minimum and maximum diameter in a pop-up window.
  - ▶ Change the filter settings as needed  
**Further information:** "Filter settings", Page 27
  - ▶ Confirm the diameter range with **OK**
  - ▶ The control loads all machining position within the selected diameter range into the List View window.







### Filter settings

After you have used the quick selection function to mark hole positions, a pop-up window appears in which the smallest diameter found is to the left and the largest diameter to the right. With the buttons just below the diameter display you can adjust the diameter so that you can load the hole diameters that you want.

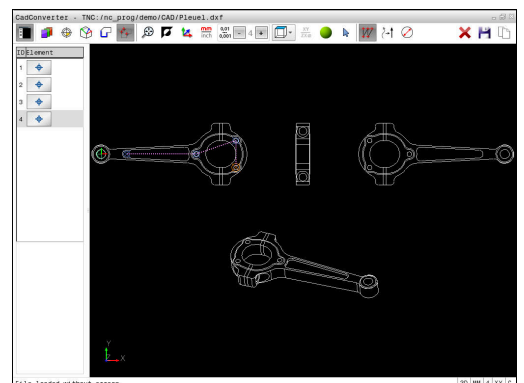
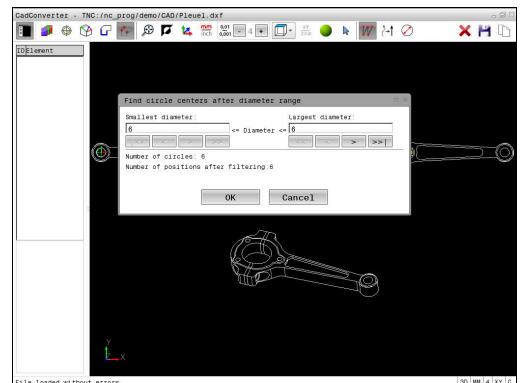
The following buttons are available:

Icon	Filter setting of smallest diameter
	Display the smallest diameter found (default setting)
	Display the next smaller diameter found
	Display the next larger diameter found
	Display the largest diameter found. The control sets the filter for the smallest diameter to the value set for the largest diameter

Icon	Filter setting of largest diameter
	Display the smallest diameter found. The control sets the filter for the largest diameter to the value set for the smallest diameter
	Display the next smaller diameter found
	Display the next larger diameter found
	Display the largest diameter found (default setting)

You can display the tool paths using the **SHOW TOOL PATH** icon.

**Further information:** "Basic settings", Page 11



### Element information

In the Element Information window, the control displays the coordinates of the last machining position selected.

You can also use the mouse to change the turning graphic. The following functions are available:

- To rotate the graphic move the mouse while holding down the right mouse button.
- To shift the displayed model, hold down the center mouse button or the mouse wheel (depending on your mouse model), and move the mouse.
- To zoom in on a certain area mark a zoom area by holding the left mouse button down
- To rapidly zoom in or out rotate the mouse wheel backwards or forwards
- To restore the standard view, double-click with the right mouse button

