Test-Drive
nanoCAD Mechanica Module

2022

General information

This tutorial contains illustrative examples of the Mechanica module capabilities in the nanoCAD Platform. The document will introduce the basic functionality of the Mechanica module to new nanoCAD users, and will deepen the knowledge of experienced users.

The test-drive consists of six sections. The first three sections are dedicated to different methods to design shafts and use/edit standard parts from the nanoCAD Mechanica library. In the next sections, users will learn how to specify parameters of drawings and work with standard fasteners.

Contents

1. Shaft Design

Launch nanoCAD with Mechanica module.

Open the ***Shaft.dwg*** file located in the ***nanoCAD Mechanica Module Test drive / Shaft Design*** folder.

1.1. In the ***Design*** group, click ***Create shaft*** (Fig. 1).

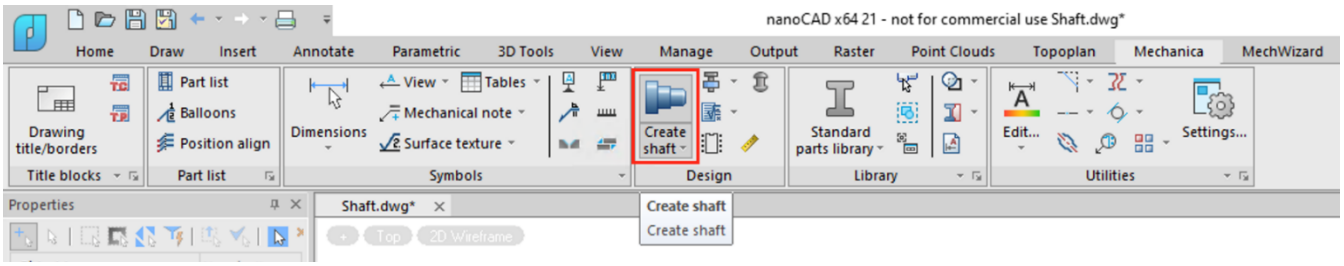


Fig. 1. The ***Create shaft*** button

1.2. You will see a dialog box and an arrow that indicates the direction in which the shaft segment is drawn. The command line displays the ***Select insertion point*** message.

- Specify the origin point for the shaft by entering coordinates in the command line (0,0). Press ***Enter***.
- The command line displays the following message: ***Select direction***.
- Specify the direction of the shaft axis by using the ORTHO snap: move the mouse cursor to the right of the center of coordinates and press ***Shift*** + left-click (Fig. 2).

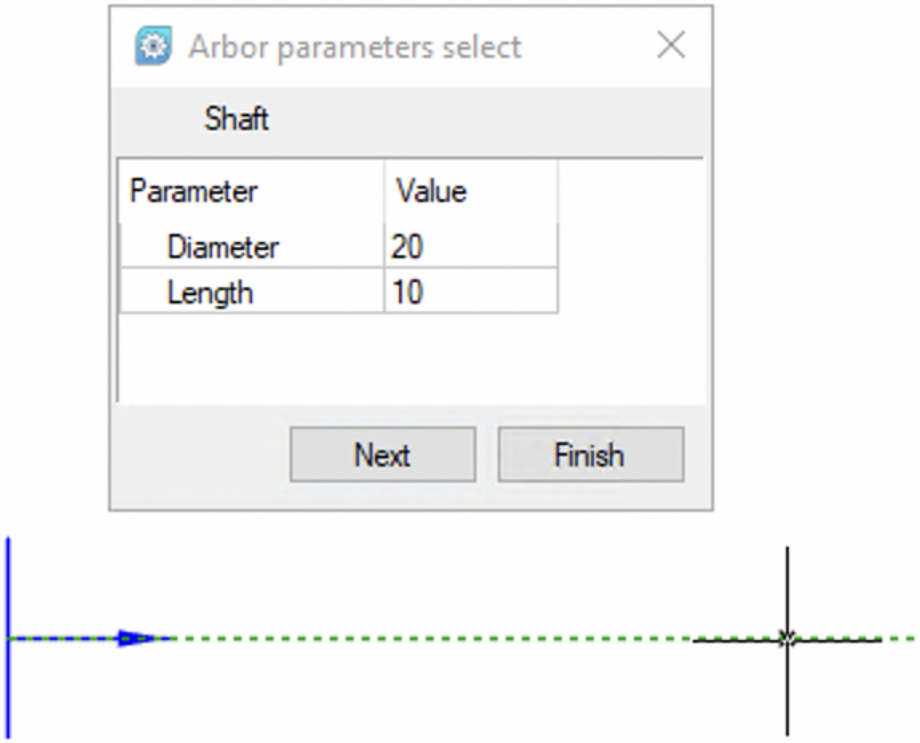


Fig. 2. Selecting the direction of the shaft axis

1.3. As you move the cursor, you will see how the diameter and length of the shaft section change in the ***Arbor parameters select*** dialog box (Fig. 3). Set the value of the shaft diameter 60 and length 10 by moving the cursor. You can move between the fields of values using the ***up*** and ***down*** keys. You can enter values in the digital keyboard, if it is more convenient. Press ***Next*** or just left-click in the model space, if the values are set by moving the cursor, to confirm.

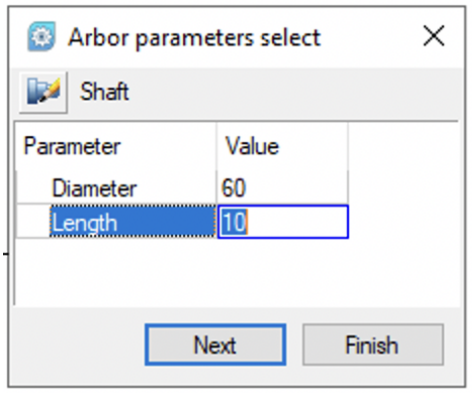


Fig. 3. The ***Arbor parameters select*** dialog box

1.4. After that, the command for drawing shaft sections will be resumed. The origin point of the next shaft section is the end point of the previous section. Draw the second section of the shaft. Enter 40 and 50 in the ***Diameter*** and ***Length*** text boxes. Confirm by clicking ***Next***. The result is shown in Fig. 4.

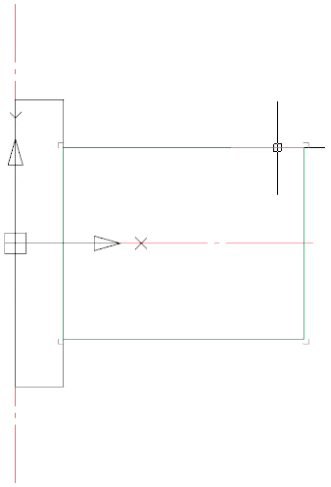


Fig. 4. Shaft with two sections

1.5. Click ***Shaft*** in the ***Arbor parameters select*** dialog box. Select the ***Conical shaft end DIN 1448*** in the drop-down menu (Fig. 5).

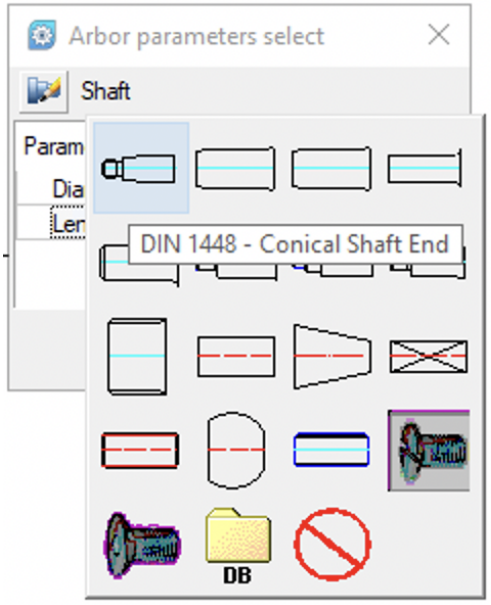


Fig. 5. Selecting a standard shaft end

1.6. Move the cursor to set the largest diameter of the shaft end to make it equal to the diameter of the cylindrical section. Use object snaps to do it (Fig. 6).

Left click to complete the shaft drawing. The centerline is drawn automatically. The drawn object is a one-piece construction.

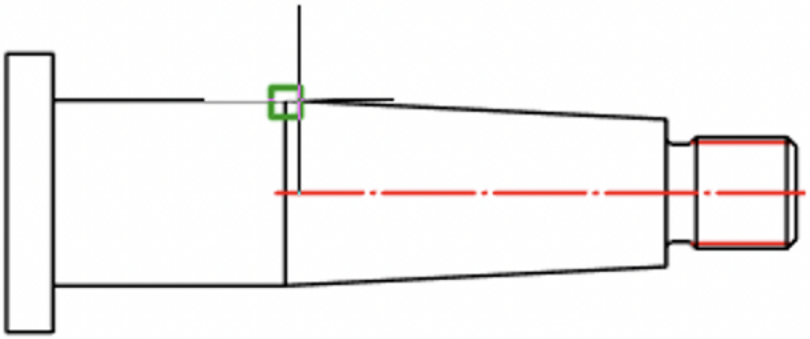


Fig. 6. Setting the largest diameter of the standard shaft end

1.7. Turn off the visibility of the ***AUXILIARY*** layer on the ***Properties*** bar by selecting it in the list of layers and clicking on the light bulb icon (Fig. 7).

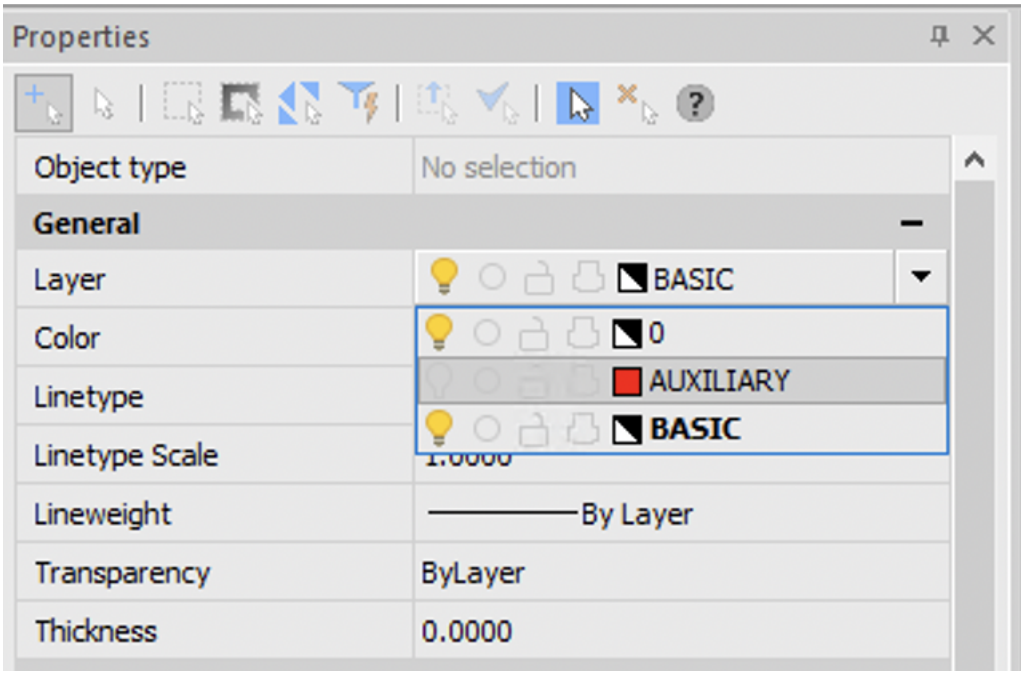


Fig. 7. Turning off layer visibility

Save the file to the ***Exercises*** folder.

2. Shaft Editing

Open the ***Shaft end.dwg*** file in nanoCAD Mechanica, located in the ***nanoCAD Mechanica Module Test drive / Shaft editing*** folder.

2.1. Start the shaft end editing with a double click on its contour.

In the ***Properties*** tab of the dialog box that appeared, change the parameters provided by the relevant standard for the shaft end (Fig. 8). Click ***Apply*** to see the changes.

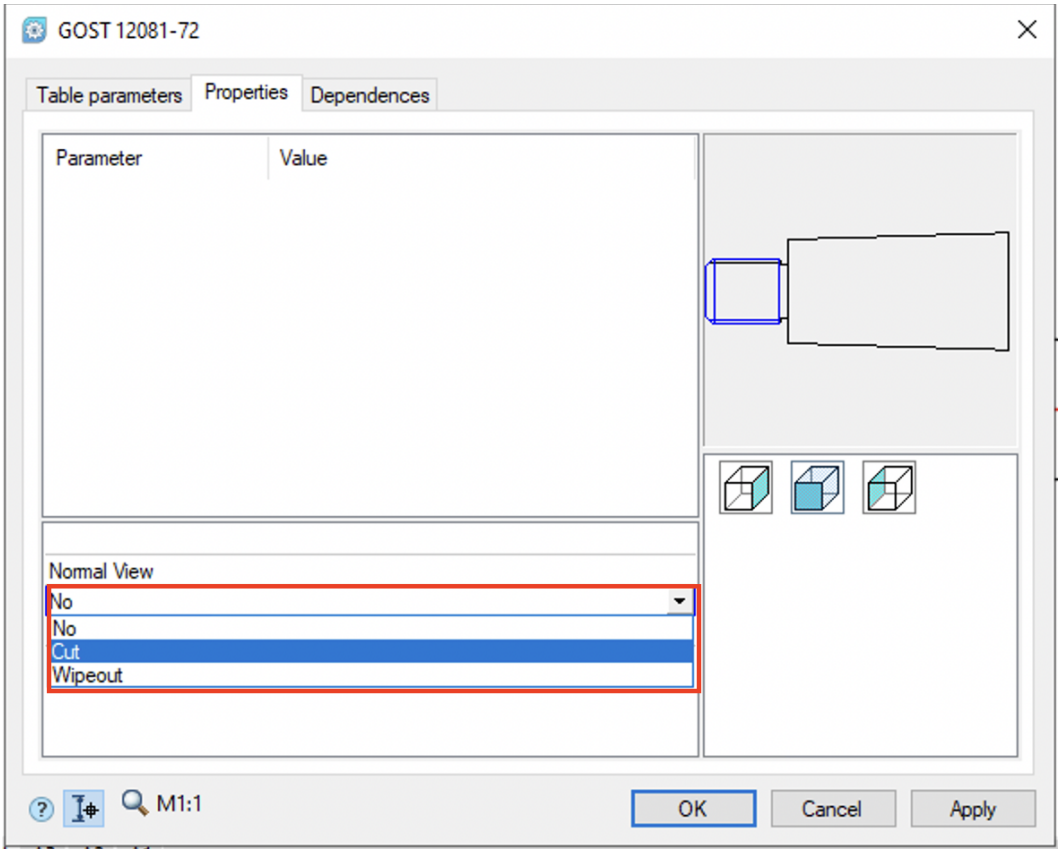


Fig. 8. Changing shaft end parameters

Leave the default properties and close the dialog.

2.2. Now, let's edit the first section of the shaft.

Double left-click on the axis (Fig. 9). In the ***Edit Shaft*** dialog box you will see the sequence for creating the shaft segments. When you select a segment, its geometry will be highlighted. Click the diameter field of the first shaft section and enter the value 45 on the keyboard (Fig. 10). Do not close the dialog.

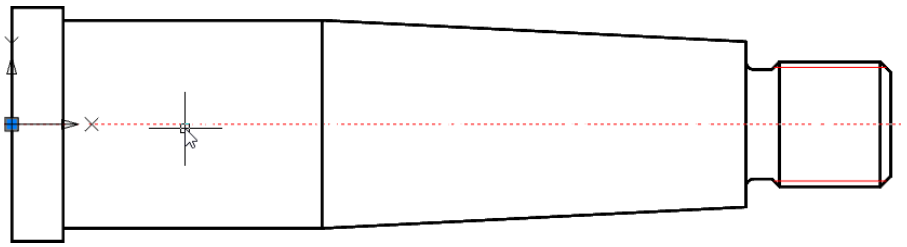


Fig. 9. Resizing a shaft section interactively

2.3. Create a gear on the shaft.

2.3.1. Click the ***MCS\Tooth-wheels*** button located in the bottom part of the window.

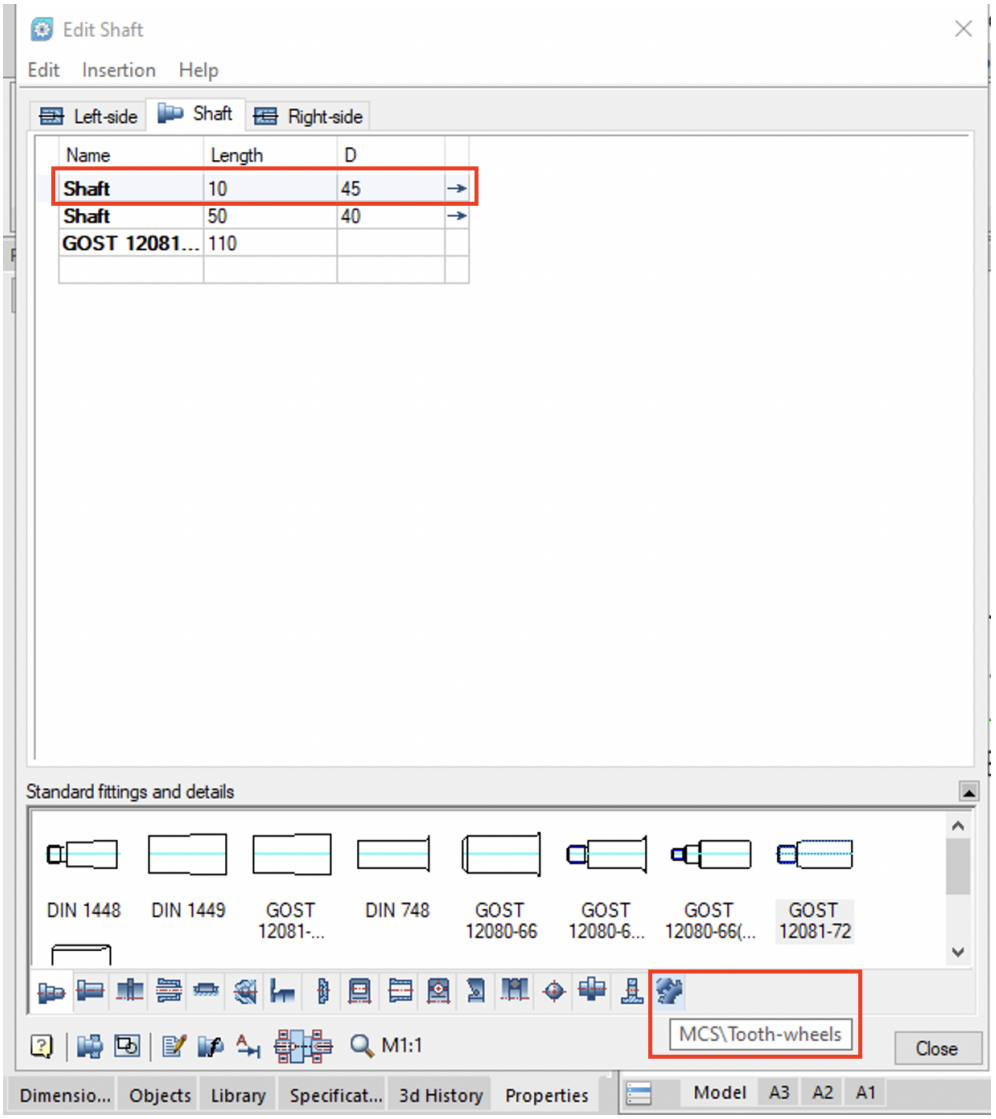


Fig. 10. ***Edit shaft*** dialog

2.3.2. Select ***Spur pinion*** (Fig. 11) from the elements that appear.

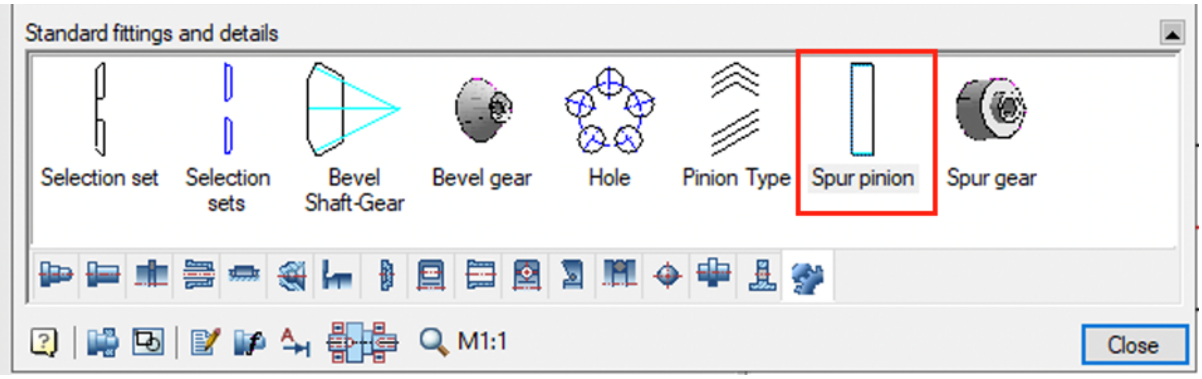


Fig. 11. ***Spur pinion*** element

2.3.3. Pick a point to insert the gear. Move the cursor to the first section of the shaft.

The area is highlighted in green to indicate that the pinion is automatically connected to the shaft (Fig. 12). The created object will be one-piece. Left-click to confirm the insertion point.

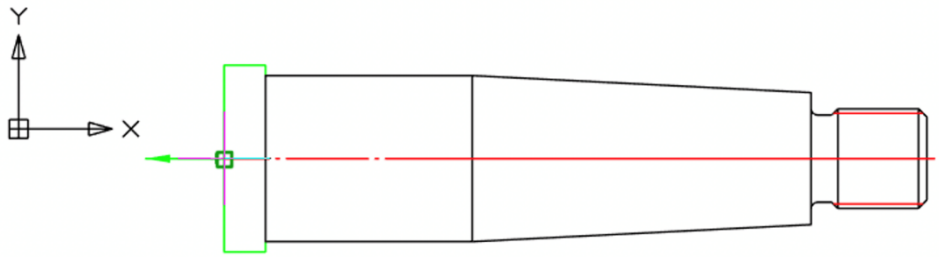


Fig. 12. Picking the insertion point for the pinion

2.3.4. You will see the gear parameters in the **Spur *gear*** dialog (Fig. 13.).

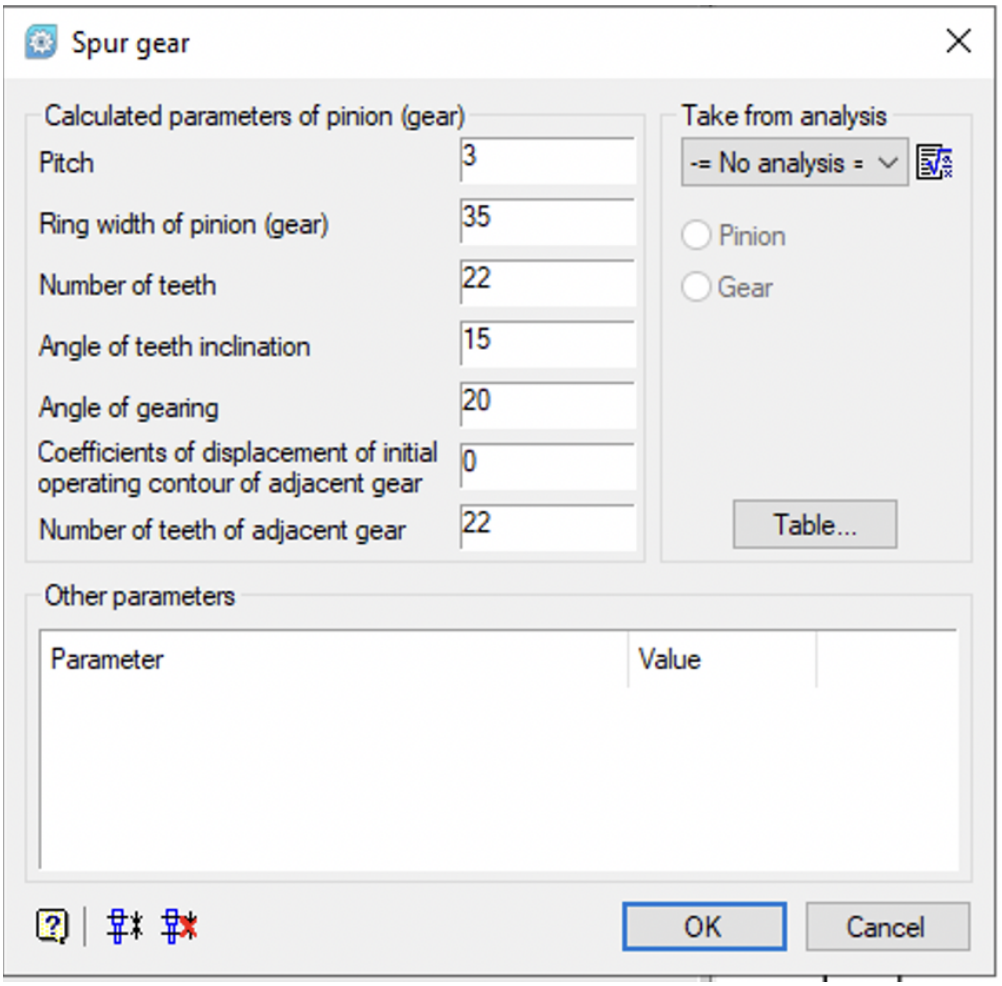


Fig. 13. The ***Spur gear*** dialog

2.3.5. Click ***OK*** to confirm. The gear model is drawn now (Fig. 14). The command to create a gear remains active. Right click or press ***Esc*** on the keyboard.

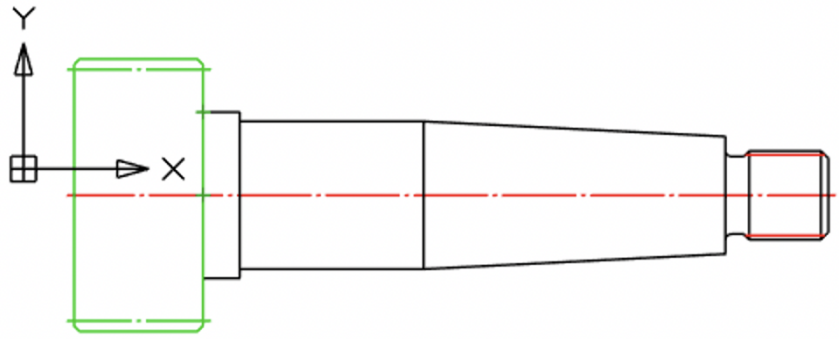


Fig. 14. Shaft with gear

2.4. Create two cylindrical shaft sections to the left of the gear. Use the dimensions shown in Table 1. Create a chamfer in the second section.

Table 1

Parameters	Diameter	Length
First section	45	10
Second section	40	20

In the ***Shaft Segments*** section select the ***Shaft*** element (Fig. 15).

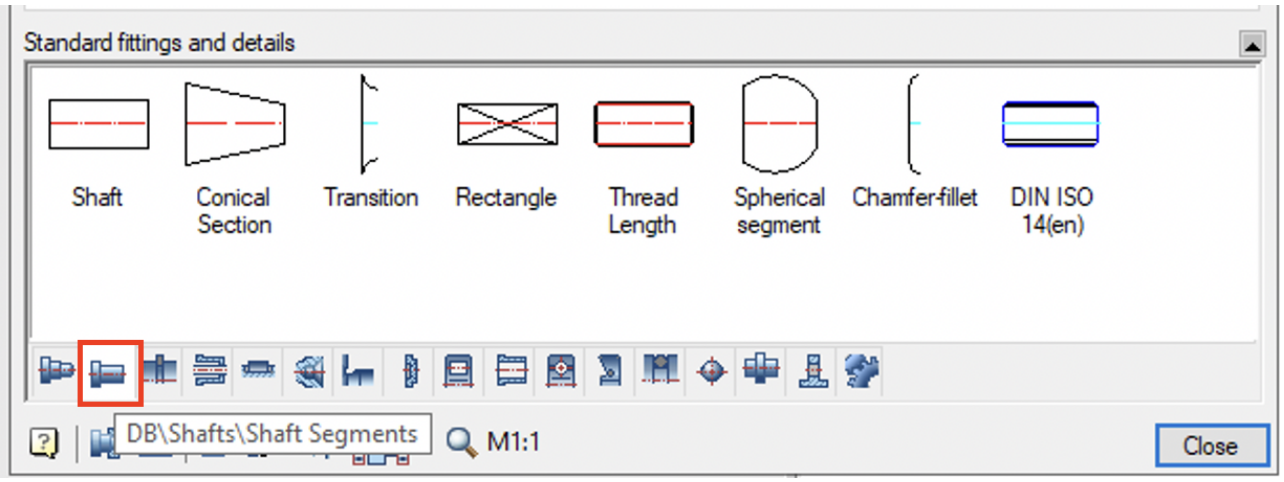


Fig. 15. *Shafts* command in the ***Edit Shaft*** dialog box

New sections of the shaft are automatically snapped to the model when you move the cursor over an existing section.
Right-click and select the ***Dialog*** command from the context menu to set the dimensions of the sections. In the dialog box, enter the required values, and click ***OK*** (Fig. 16).

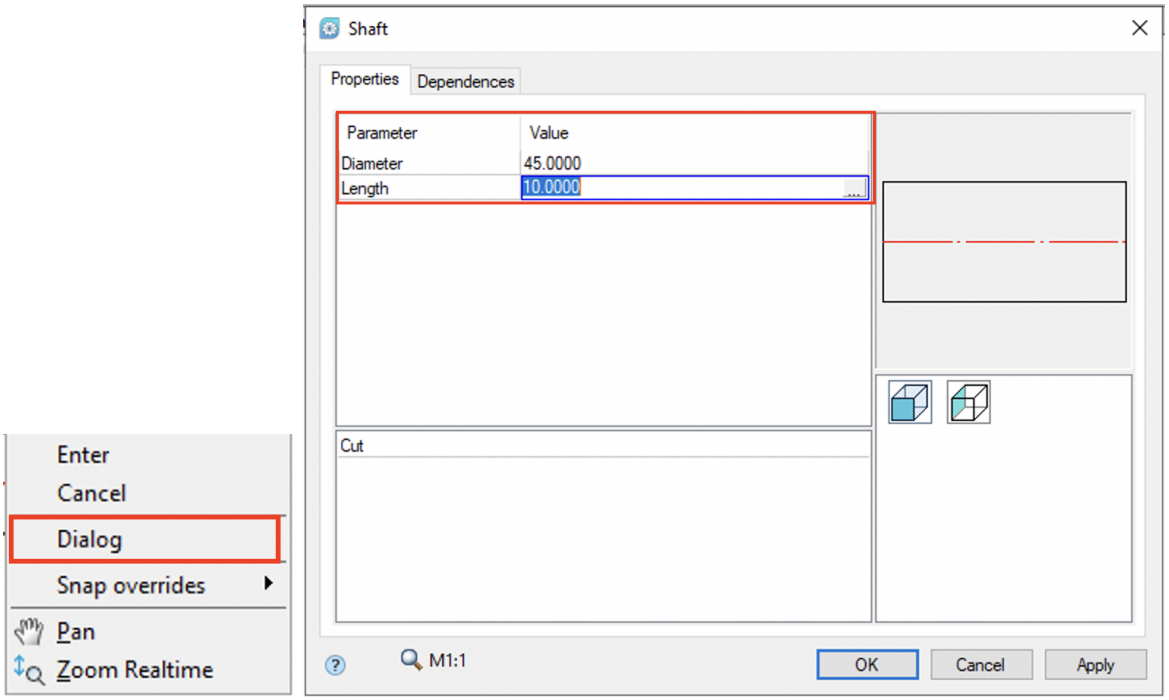


Fig. 16. Entering shaft section values in the ***Shaft*** dialog

In order to form a chamfer, specify the section in the *Shaft Edit* window and select the ***Chamfer-fillet*** element in the ***Shaft segments*** section (Fig. 17).

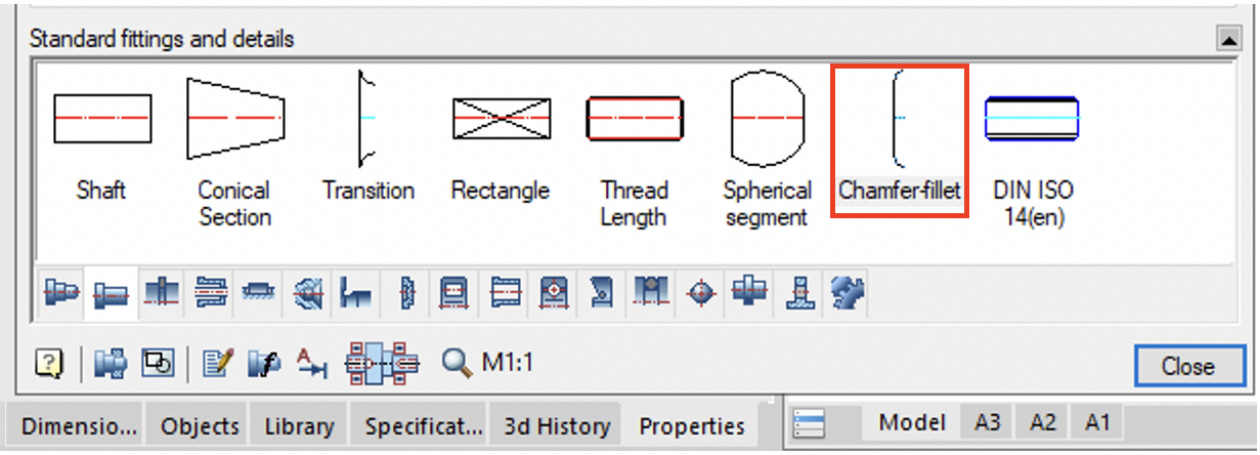


Fig. 17 The ***Chamfer-fillet*** command

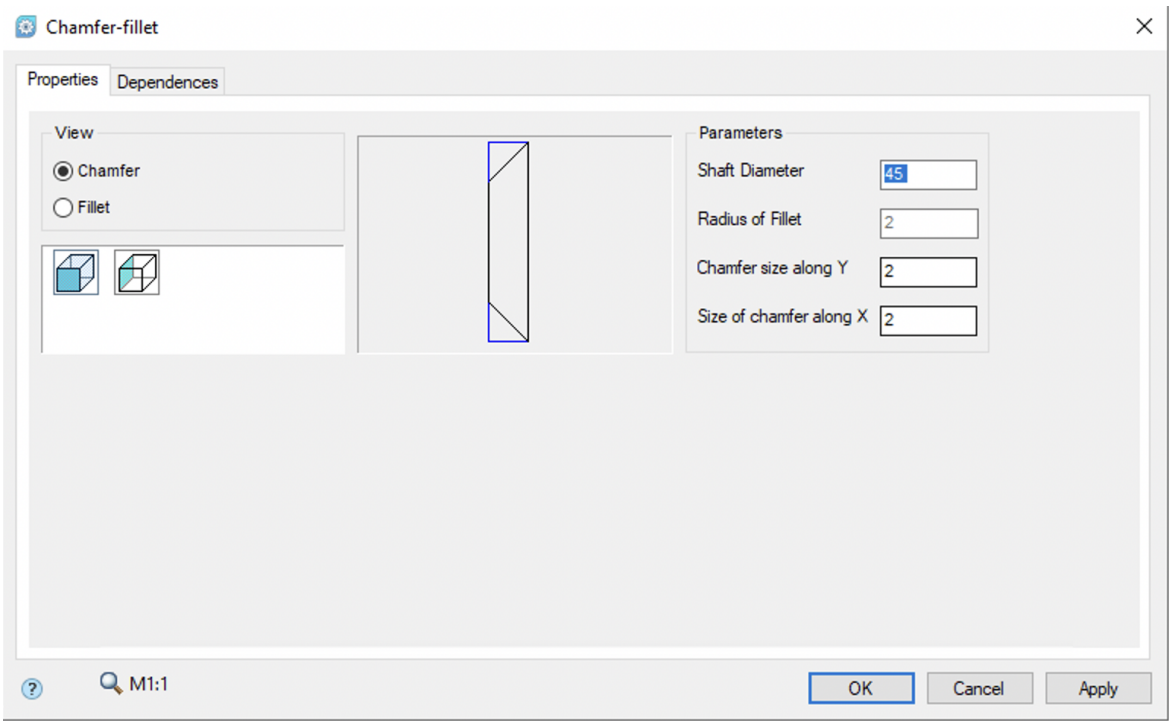


Fig. 18. ***Chamfer-fillet*** dialog box

Specify the end face of the section with a chamfer, leave the default values, and click ***OK*** (Fig. 18). Cancel the command by pressing ***Esc***. Close the shaft editing window. The result is shown in Fig. 19.

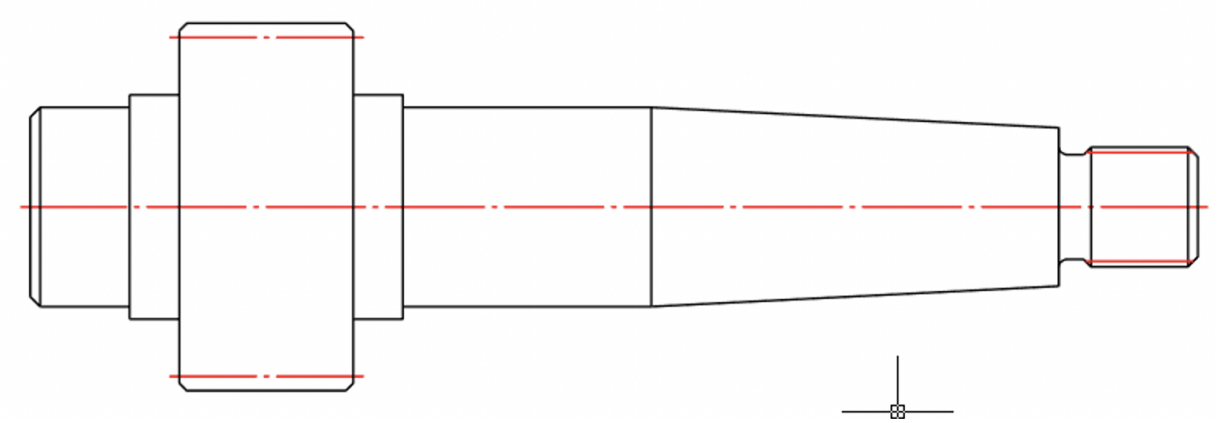


Fig. 19. The edited shaft model

Save the file to the ***Exercises*** folder.

3. Adding Standard Elements

Open the ***High-speed shaft.dwg*** file in nanoCAD Mechanica located in the ***nanoCAD Mechanica Module Test drive / Standard parts folder***.

3.1 Double-click on the centerline to open the ***Edit Shaft*** dialog box. In the ***DB/Shafts/Ball bearings*** section (click the icon in the bottom of the dialog) select the bearing GOST 8338-75 (fig. 20).

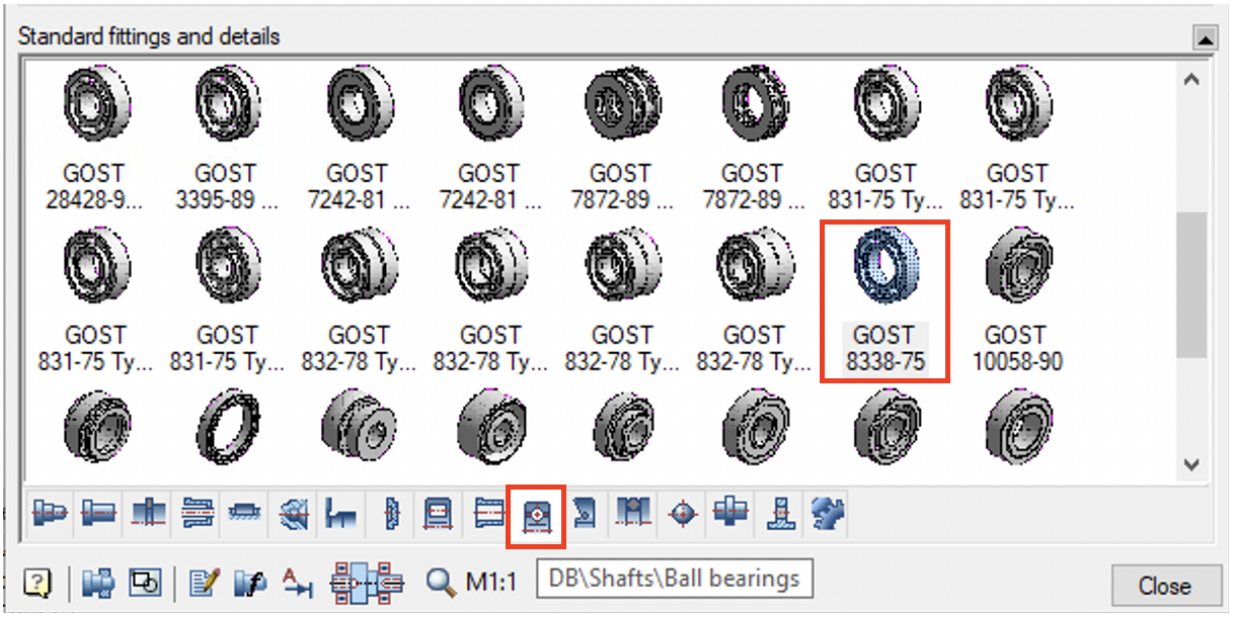


Fig. 20. GOST 8338-75

Set the bearing position as shown in fig. 21: move the cursor to the intersection of the first section's end line and the surface of the second one. After the ***Endpoint*** snap is triggered. Click to fix the bearing position. The inner diameter of the standard part will be determined automatically.

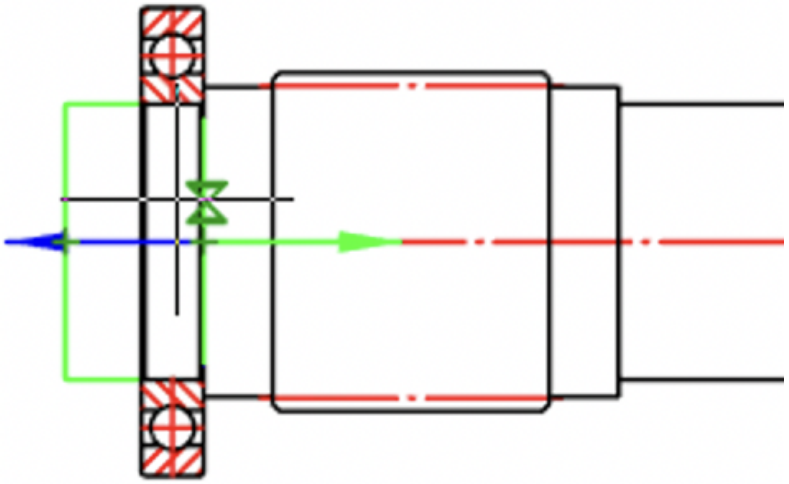


Fig. 21. Placing of the bearing on the shaft

A new dialog box with the name of the bearing appears (in this case, GOST 8338-75). Set the dimensions on the ***Properties*** tab: bearing view (frontal or side). If you change the inner diameter of the bearing, the diameter of the corresponding shaft section will also change. The ***Dependences*** tab shows the parametric and geometric dependences applied to the bearing. In the ***Calculation*** tab you can calculate the bearing longevity for given radial and axial loads.

Make sure the ***Dynamic input*** option is disabled. Set the bearing dimensions as shown in fig. 22 and click **OK**.

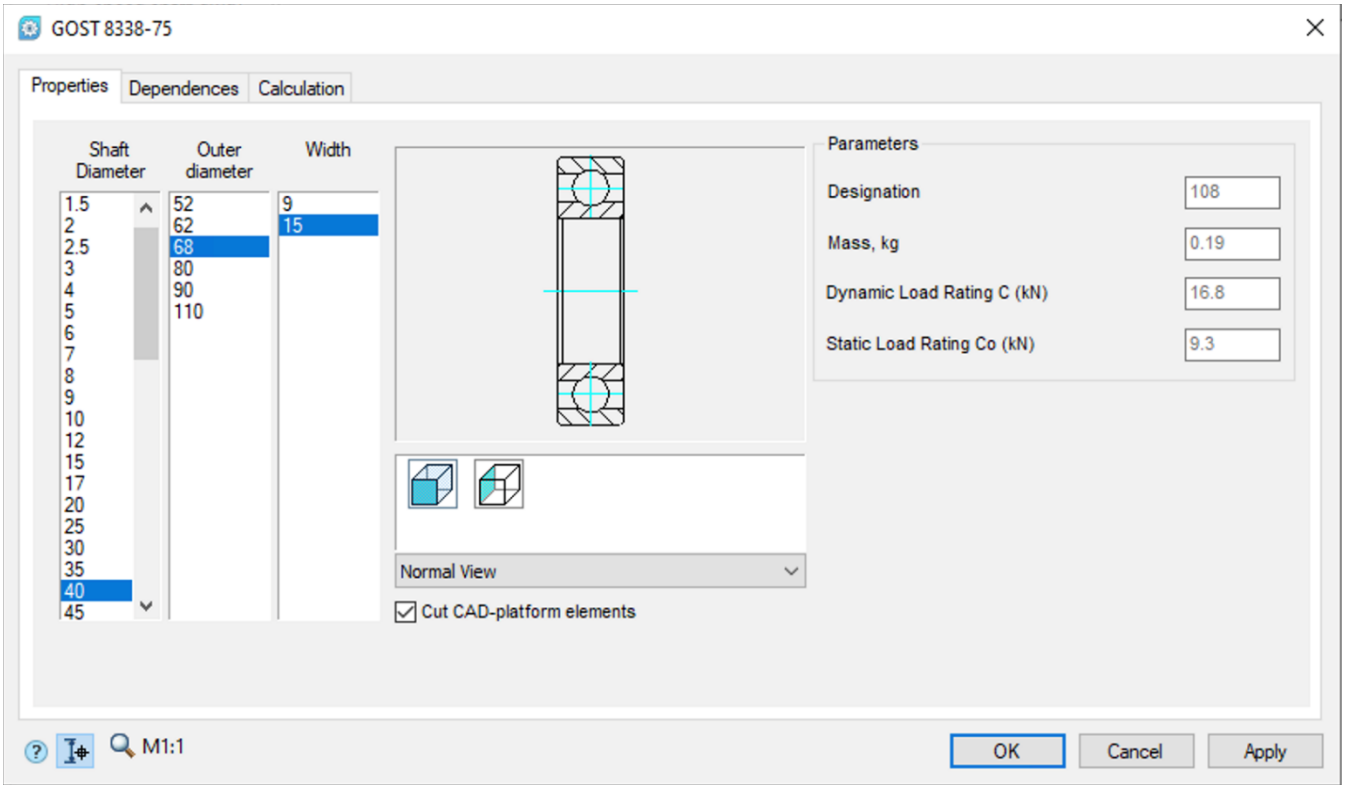


Fig. 22. Dialog box *GOST 8338-75*

Place the second bearing with the same dimensions as shown in fig. 23.

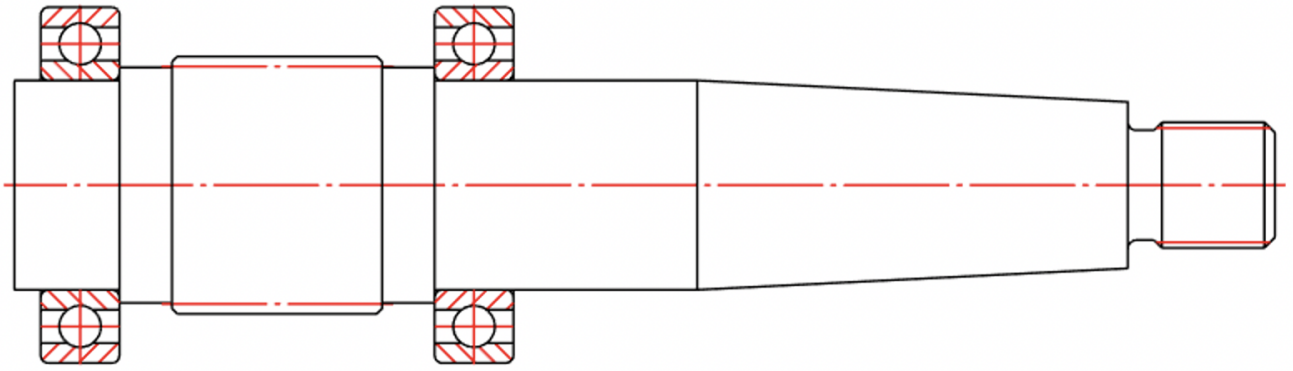


Fig. 23. Shaft with two bearings

3.2. In the ***DB/Shafts/End Caps*** section of the ***Edit shaft*** dialog box, select the cover of GOST 18513-73 (Fig. 24).

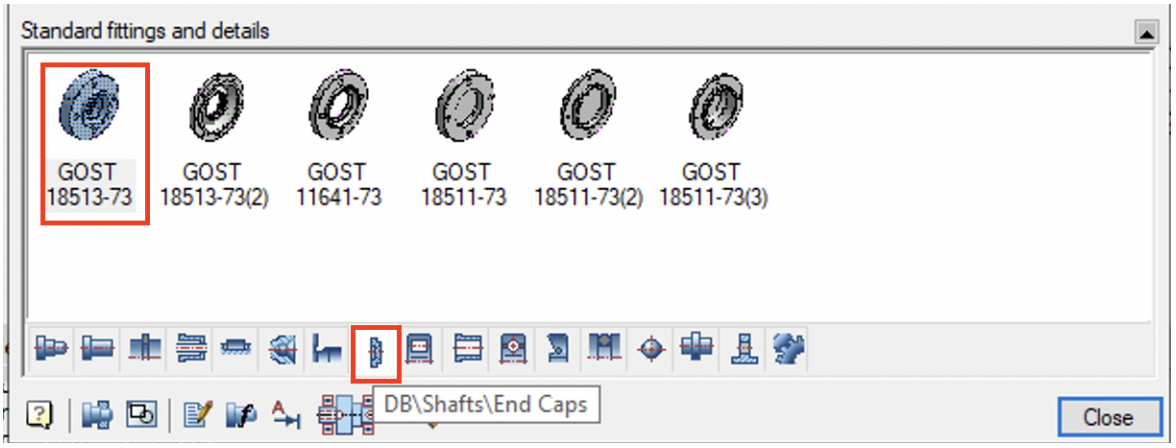


Fig. 24. End cap selection

Move the cursor to the right end of the bearing and left-click to set the cap position as shown in Fig. 25.

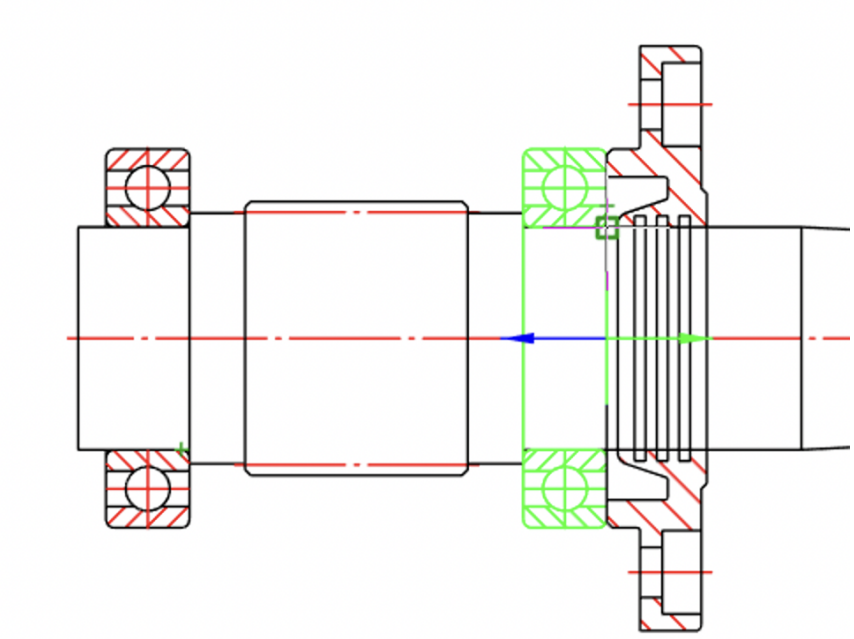


Fig. 25. Placement of the through cover

The dimensions of the cap will be determined automatically. Check the parameters in the GOST 18513-73 dialog box and click **OK** (fig. 26).

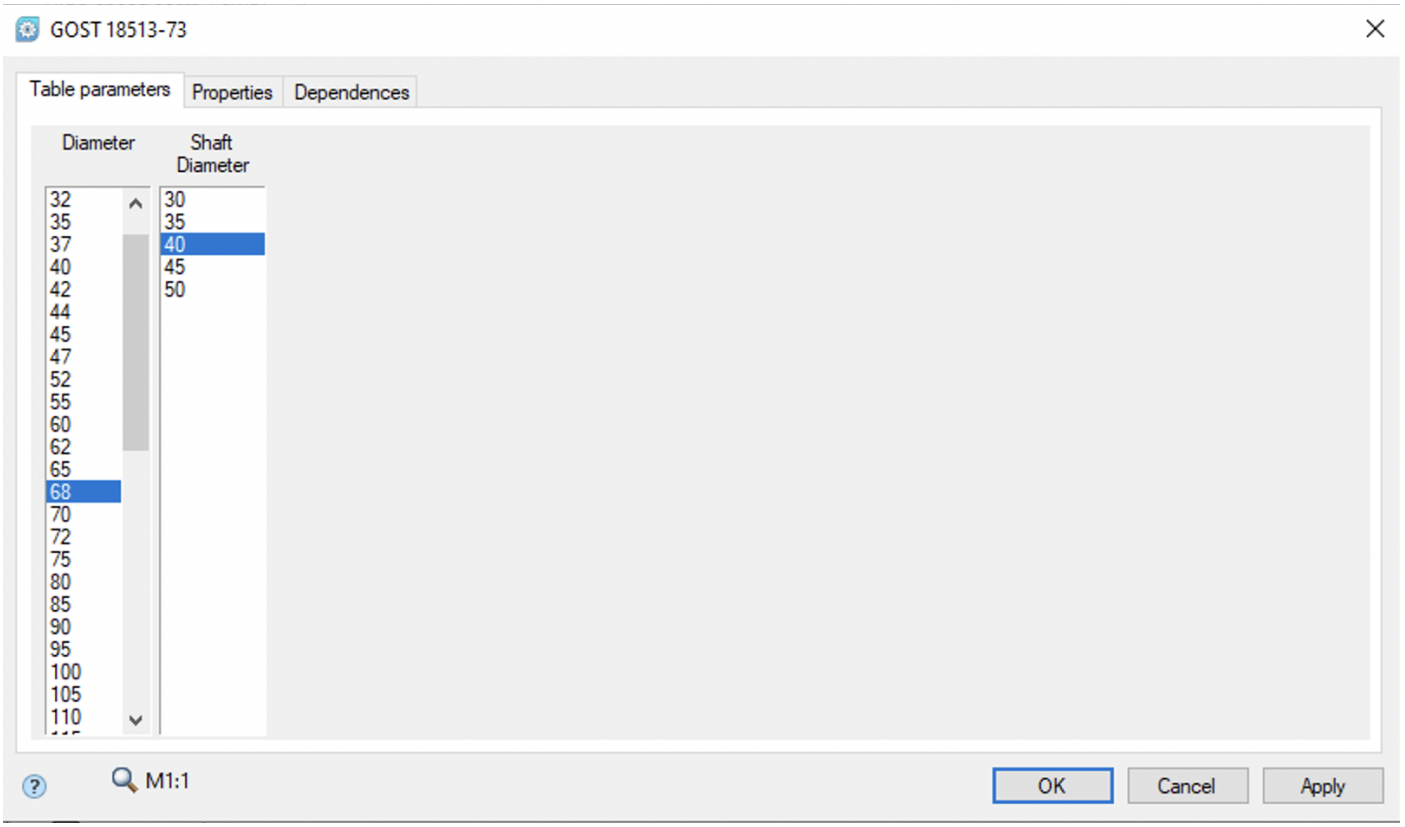


Fig. 26. Cap parameters dialog box

The cap insertion command remains active, press ***Esc*** or right-click to return to the ***Edit Shaft*** dialog box. Select the cover GOST 18511-73(2) (fig. 27) and place it as shown in fig. 28.

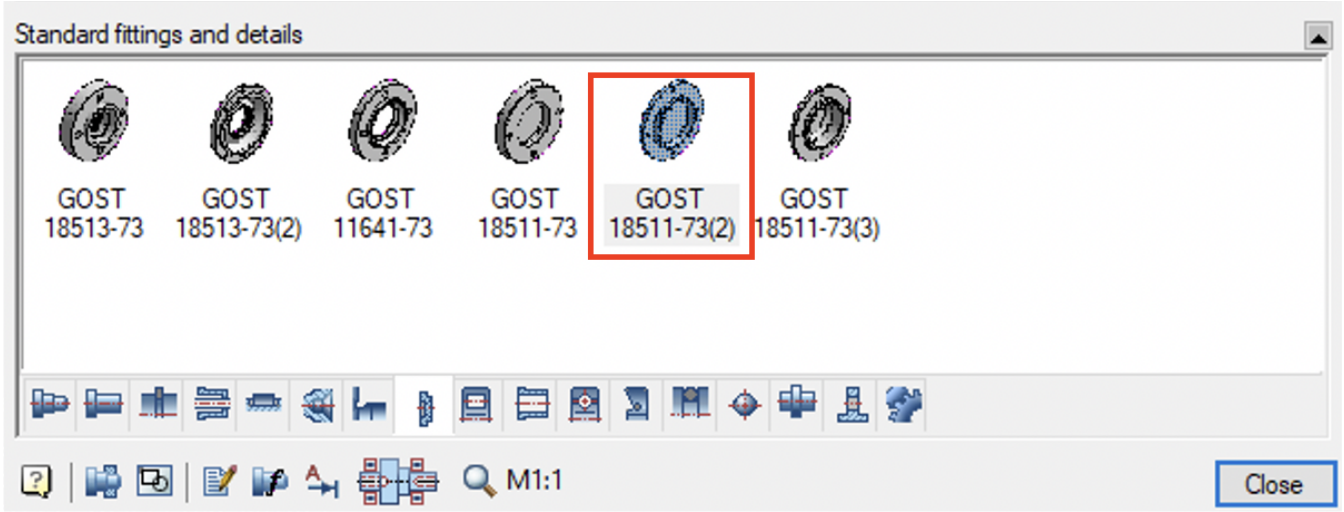


Fig. 27. Blind cover selection

Press ***Esc*** and close the ***Edit Shaft*** window.

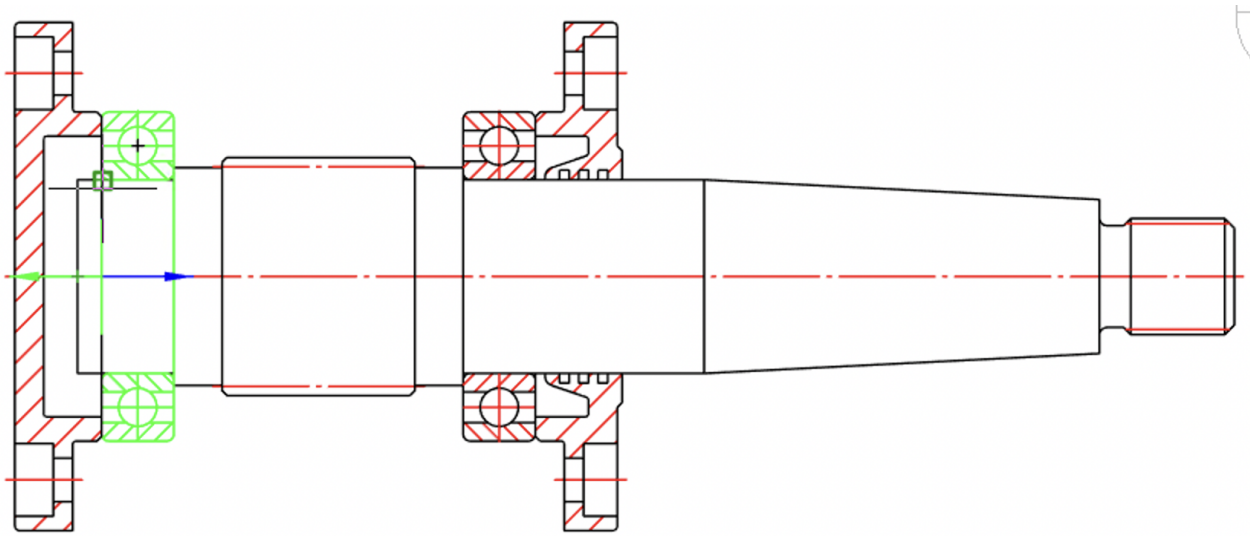


Fig. 28. Blind cover placing

Save the file to the ***Exercises*** folder.

4. Specification

Open the ***Specification.dwg*** file in nanoCAD Mechanica located in the ***nanoCAD Mechanica Module Test drive / Specification*** folder.

4.1. Positioning

Position the gear elements randomly: run the ***Balloons*** command on the ***Part list*** tab (fig. 29).

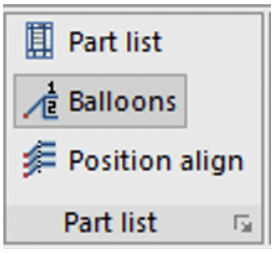


Fig. 29. The ***Balloons*** command on the ***Part list*** tab

Follow the instructions on the command line and specify the start and end points of the position (Figure 30). The start point position in shaft specifications should be defined on the axis of the part. In standard products (bearings, covers) — by indicating their contour.

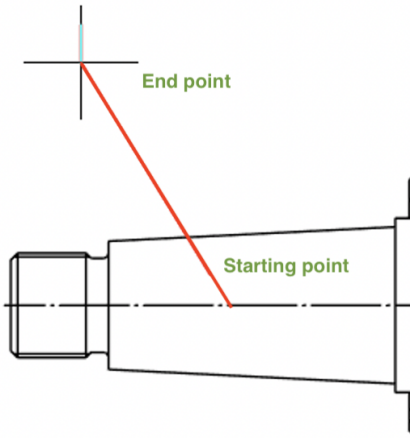


Fig. 30. Starting point and end point

In the ***Name*** field of the ***Position leader editor*** pop-up window, change the standard name to ***Driven shaft*** (Fig. 31). The ***Partition*** field will be filled in automatically. Close the window. Press the spacebar to reactivate the command.

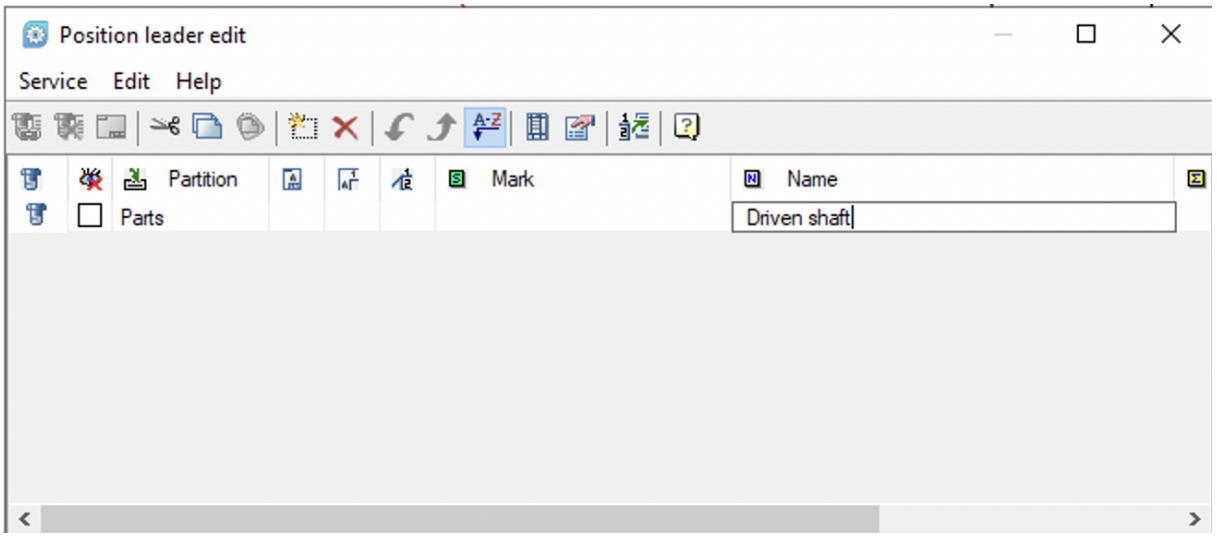


Fig. 31. ***Position leader edit*** dialog box

Note. Name the larger wheel ***Wheel*** in the specification, and name the cylindrical pinion shaft ***Leading Pinion Shaft***. No changes are required for standard products.
The final view of the gear train with placed positions is shown in fig. 32.

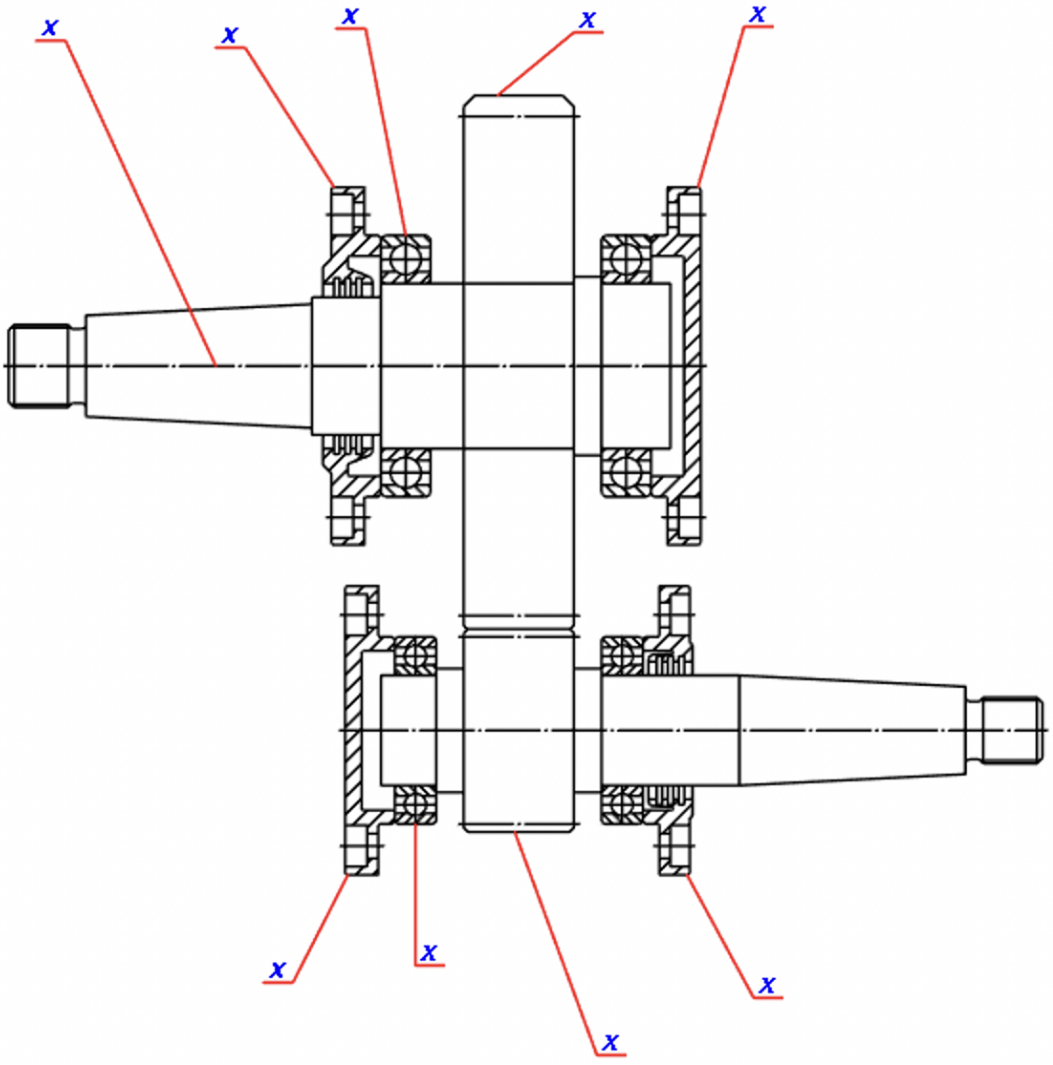


Fig. 32. The gear train with placed positions

4.2. Line position alignment

Press ***Position align*** 

on the ***Part list*** tab or via the path in the classic interface scheme: ***Mechanica*** → ***Part list*** → ***Position align*** drop-down menu. Specify the leaders of the low-speed shaft: use the frame from right to left for faster selection (Fig. 33). Press ***Enter***.

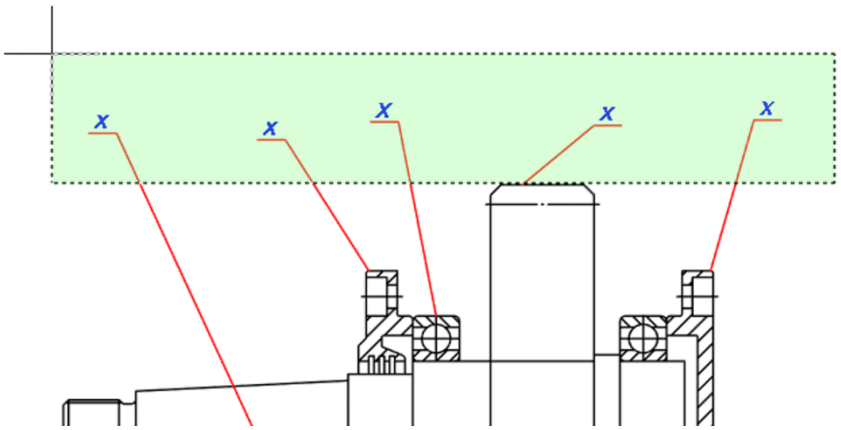


Fig. 33. Selected items

Type *L* in the command-line interface and press ***Enter***, or select ***Line*** in the context menu (Fig. 34). Specify the start and end points to position the line in alignment with the leaders.

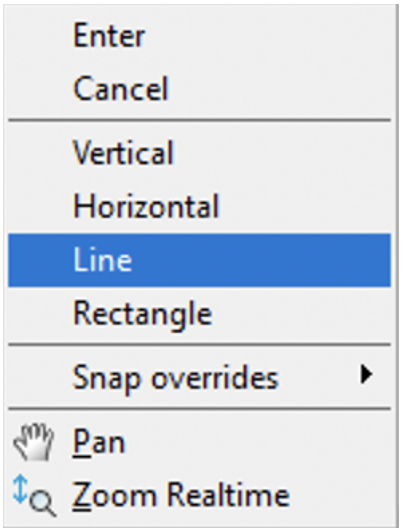


Fig. 34. The context menu of the ***Position align*** command

Align the high speed shaft positions in the same way. The final view of the gear train with placed and aligned positions is shown in fig. 35.

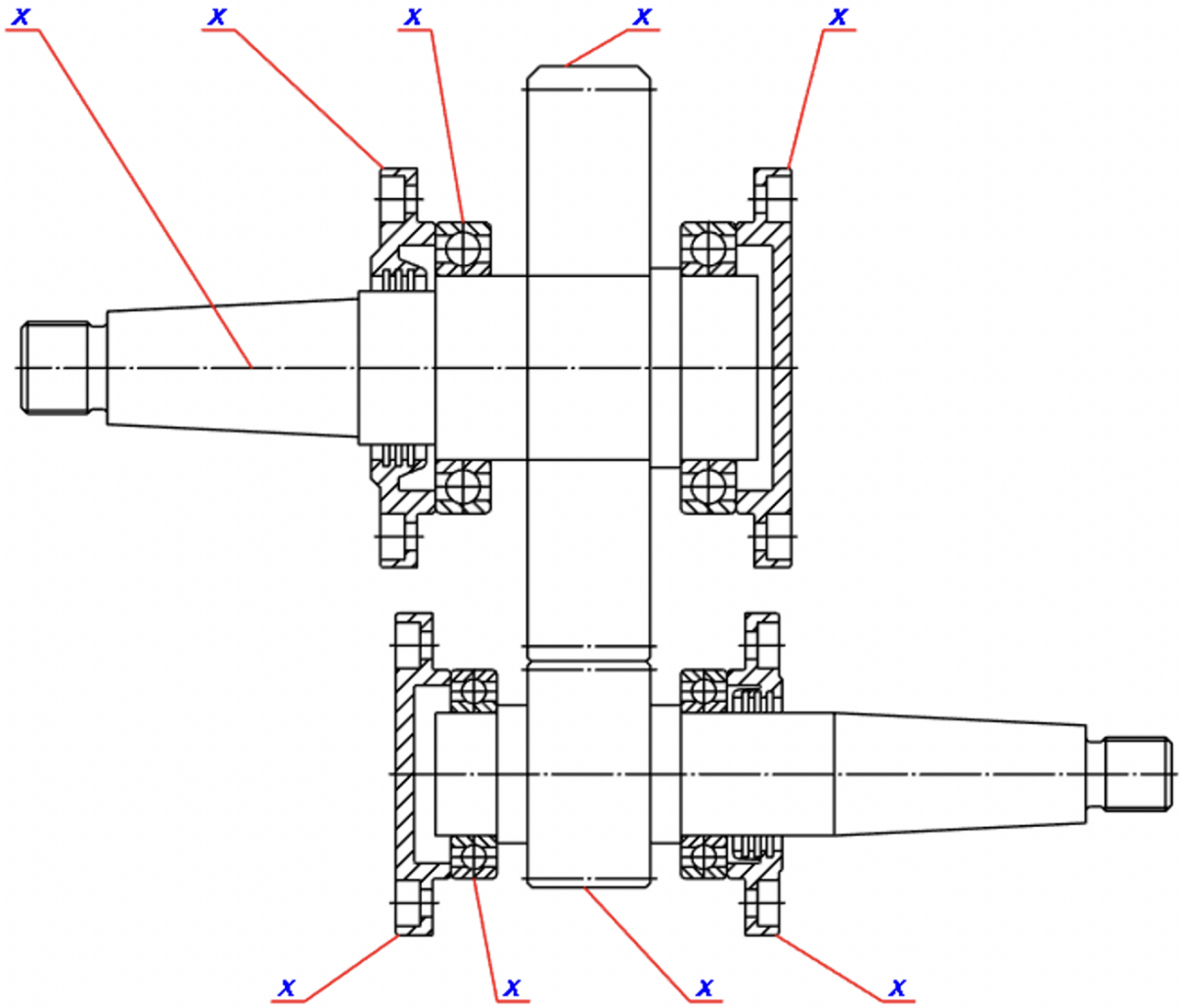
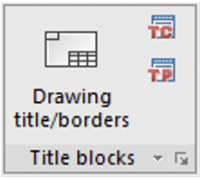


Fig. 35. The gear train with placed and aligned positions

4.3. Format and title block of the drawing.

Create format and fill in the title block: switch from model space to sheet A2 

and click ***Drawing title/borders*** on the ***Title blocks*** tab, 

. In the ***Format*** window select A2 and click ***OK*** (Fig. 36). Place the format appeared on the sheet at the origin with the End point anchor.

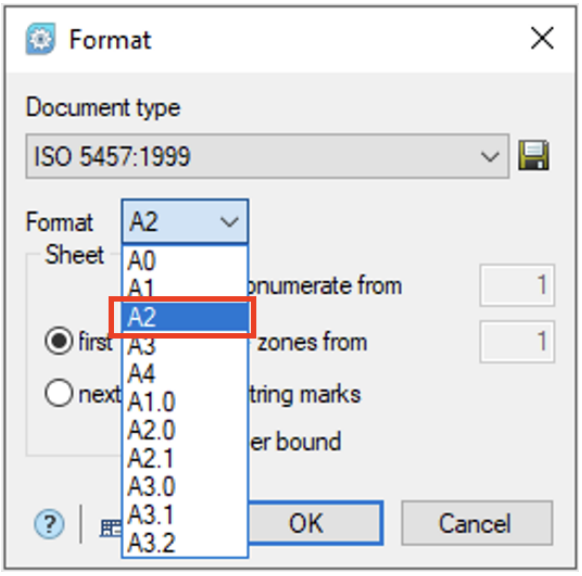


Fig. 36. Creating format

Double-click on the title block's outline to edit. In order to fill the title block fields and additional graphs, double-click on any line of the title block (Fig.37).

Press ***OK***.

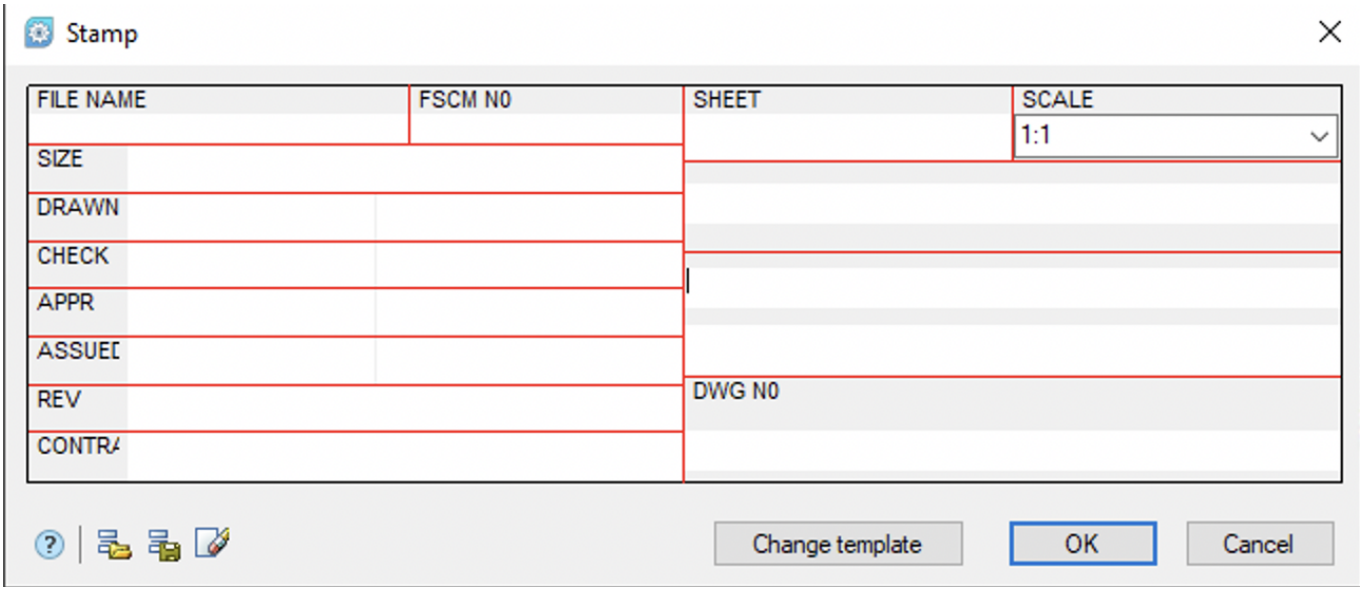


Fig. 37. Filling in the fields of the title block

4.4. Specification settings.

Go to sheet A4. Click the ***Specification edit*** button on the ***Specification*** tab or via the path: ***Mechanica*** → ***Part list*** → ***Part list*** drop-down menu (in the classic interface scheme).

In the *Specification edit* dialog box, click on the ***Assembly unit*** list on the left side. Double-click ***Parts*** and sort the names in alphabetical order with the ***Sort*** button (Fig. 38). Repeat the actions for ***Standard parts***.

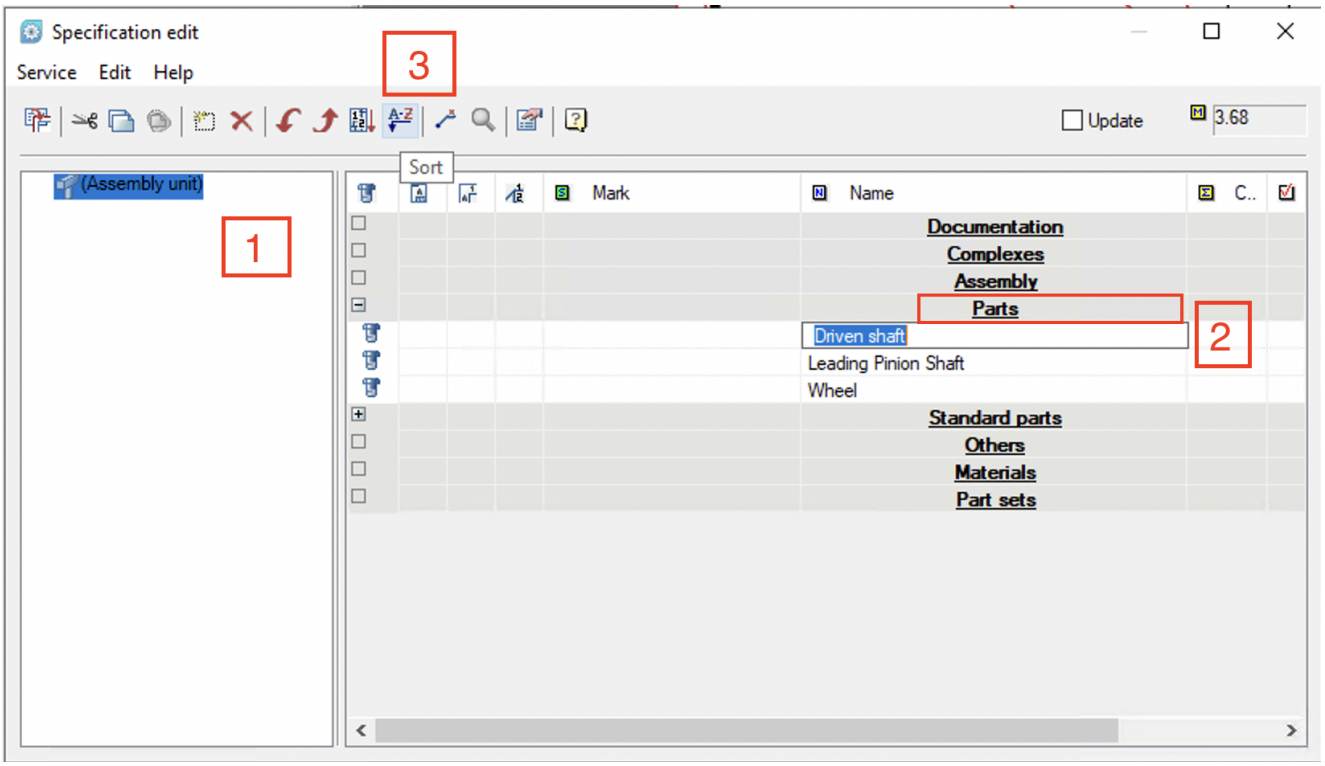


Fig. 38. Sorting titles alphabetically

In the ***Specification edit*** dialog box, click the ***Specification settings*** button (***Service*** → ***Specification settings***). On the ***Numbering*** tab, set the parameters for the ***Parts and Standard parts*** sections as shown in fig. 39. Click ***OK***.

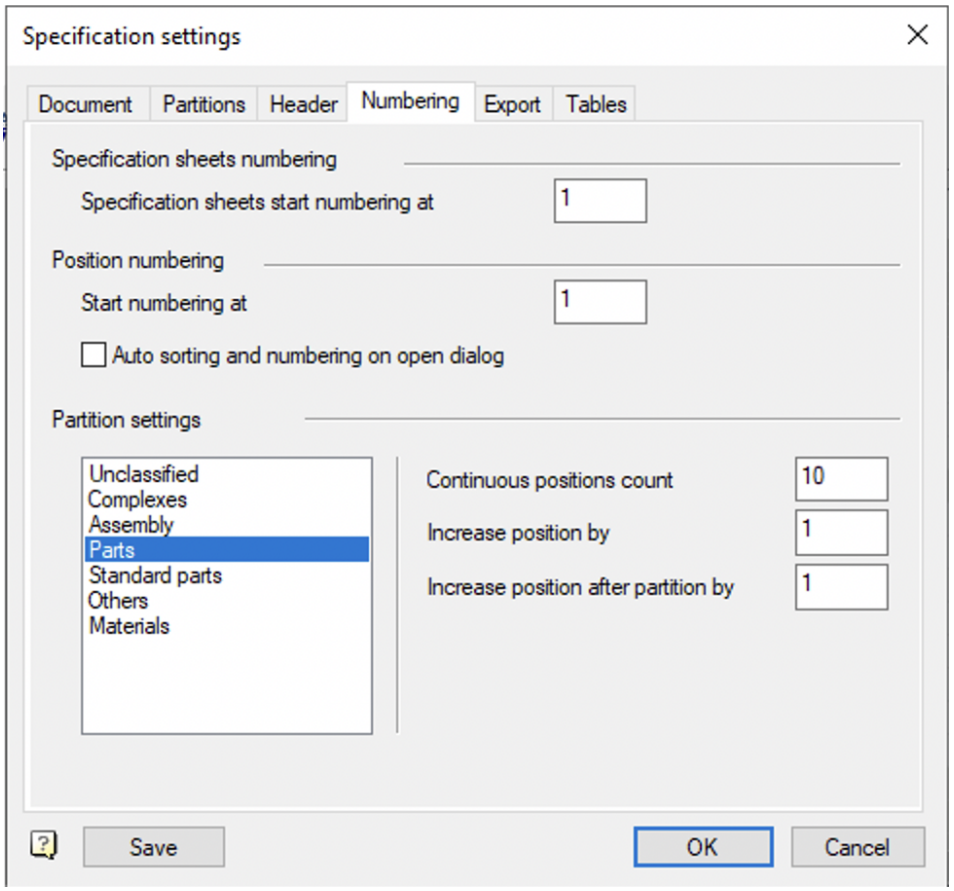


Fig. 39. Setting the numbering

Press ***Do numbering*** .
The final view of the specification window is shown in Fig. 40.

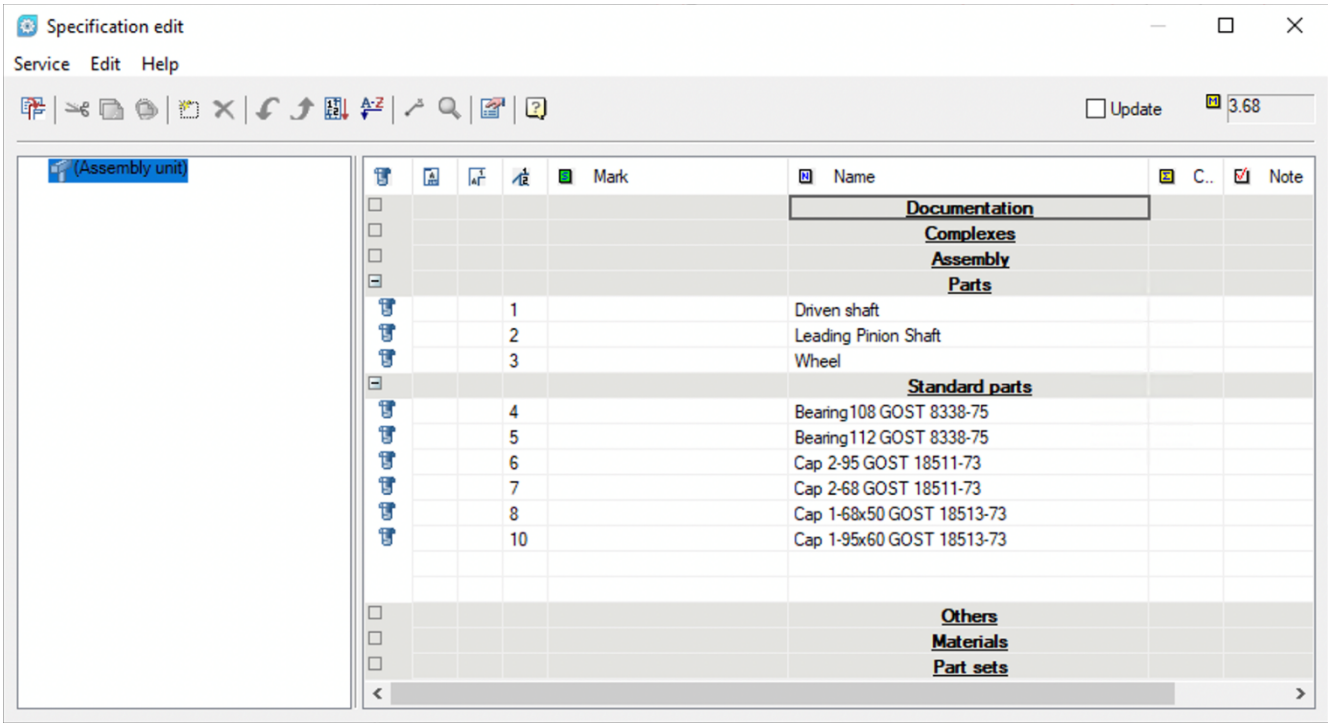


Fig. 40. Completed *Specification edit* window

Press *Export to drawing* .
Paste the generated specification into the A4 format field (Fig. 41).

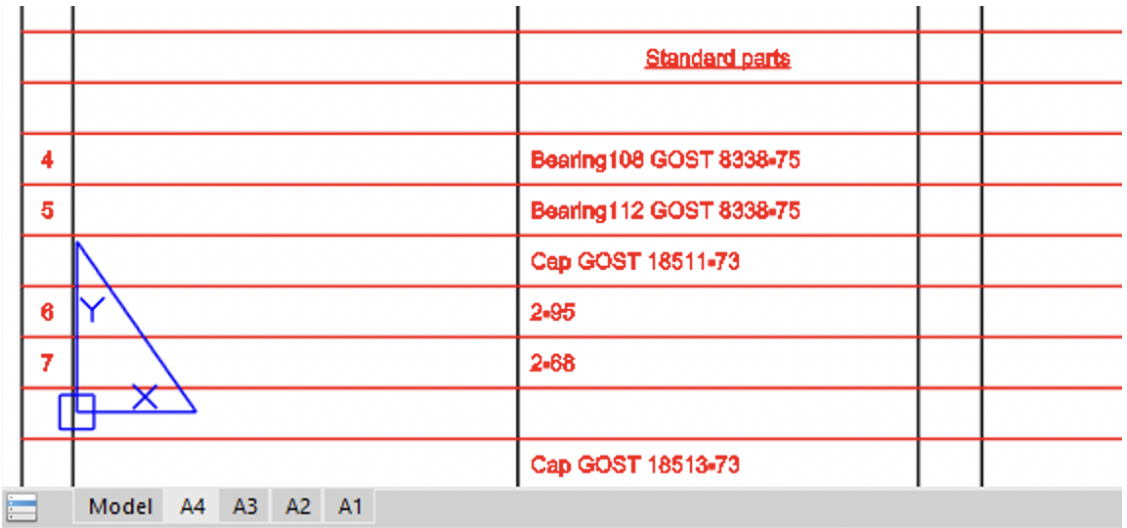


Fig. 41. Pasting the specification into the A4 format

Close the ***Specification edit*** window. Save the file to the ***Exercises*** folder.

5. Symbols

Open the ***Symbols.dwg*** file located in the ***nanoCAD Mechanica Module Test drive / Symbols* folder**.

5.1. Insert designation of bases.

Click the ***Datum identifier*** *button* on the ***Symbols*** tab or use the following path in the classic interface: ***Mechanica*** → ***Symbols*** → ***Datum identifier***. Select a point on the left edge of the shaft axis with the ***Endpoint*** anchor, as shown in Fig. 42.

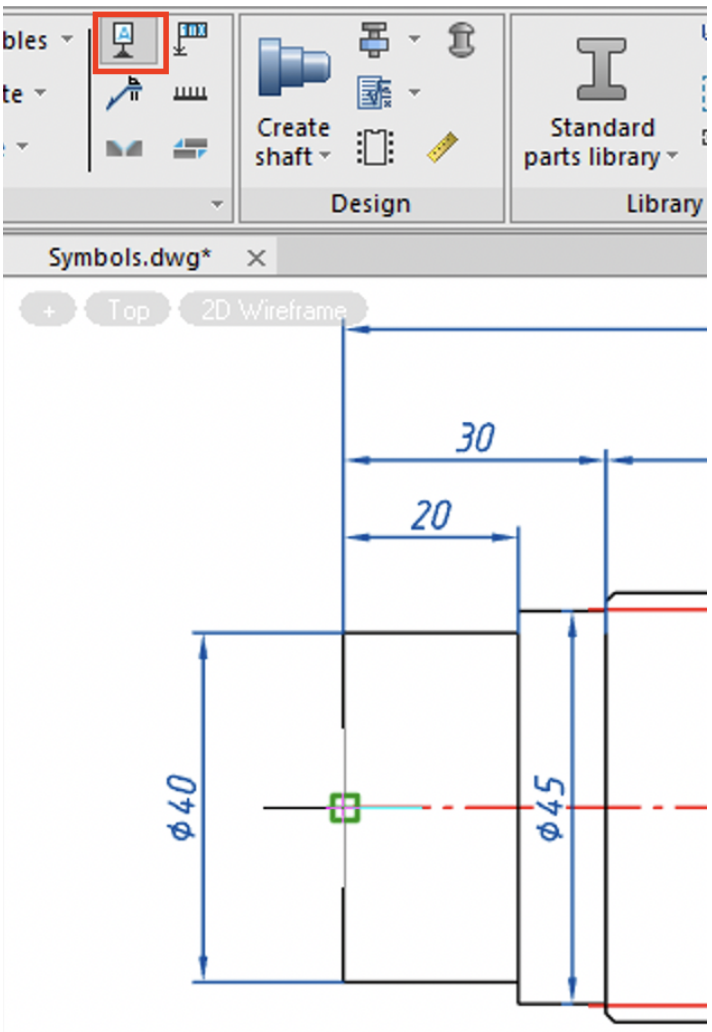


Fig. 42. Endpoint on the shaft axis to insert the datum symbol

Select the symbol position in the drawing and press ***Enter*** or left-click to confirm.

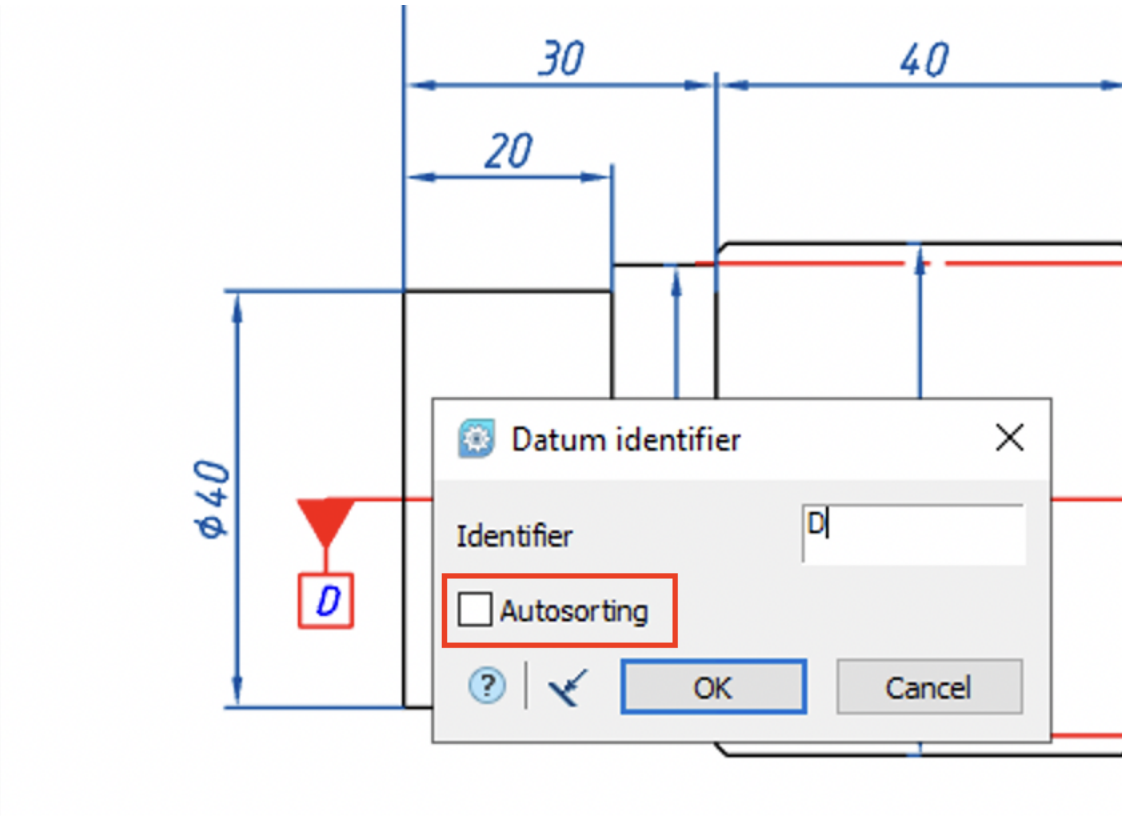


Fig. 43. Setting the designation of the base and its editing

Double-click on the base designation to open the ***Datum identifier*** window and disable the autosorting option: enter ***D*** in the ***Identifier*** field (Fig. 43) and click ***OK***.
Once the ***Autosorting*** field is activated, the base designations are sorted in alphabetical order, with the previously used letter designations taken into account.

5.2. Designation of form and position tolerances.

Enable the ***ORTHO*** mode in the status bar or press F8.

Click ***Feature control frame*** on the ***Symbols*** tab, or open from the menu in the classic interface: ***Mechanica*** → ***Symbols*** → ***Feature control frame***. Set a point on the first section of the shaft.
When you select a base surface, the graphic is highlighted in green.

Pick the next point to anchor the sign line.

Place the tolerance mark under the third section of the shaft: indicate new points in the drawing by moving the cursor (Fig. 44). If you need to undo the actions, use the ***Back*** command from the context menu or from the command line.

Press ***Enter*** to finish setting the location.

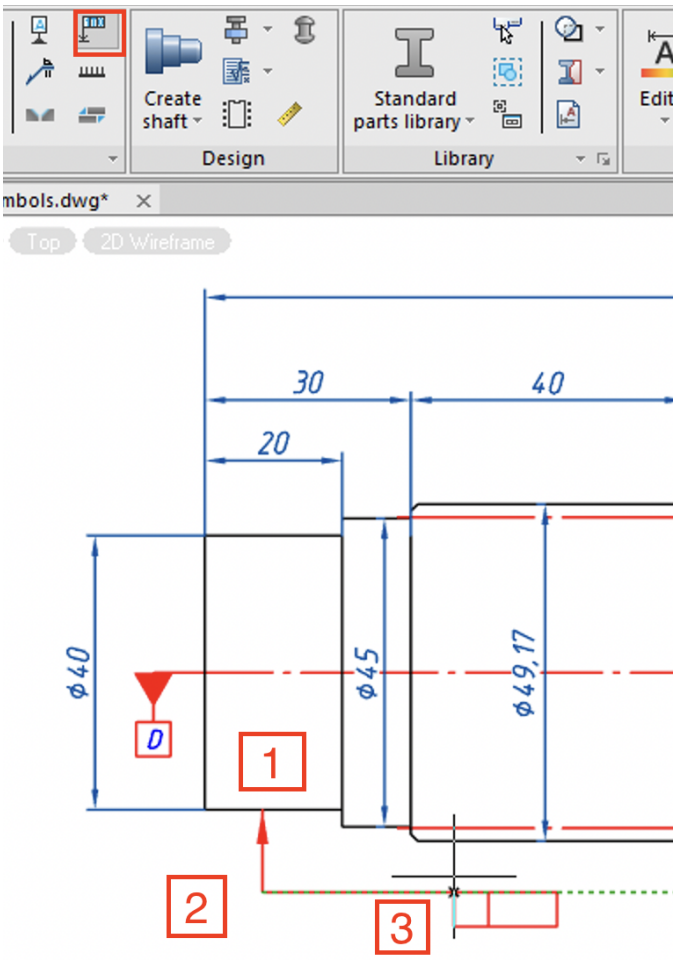


Fig. 44 Setting designation of form and location tolerances

Set ***Coaxial tolerance*** in the first field of the ***Form&plane position tolerance*** window (Fig. 45).

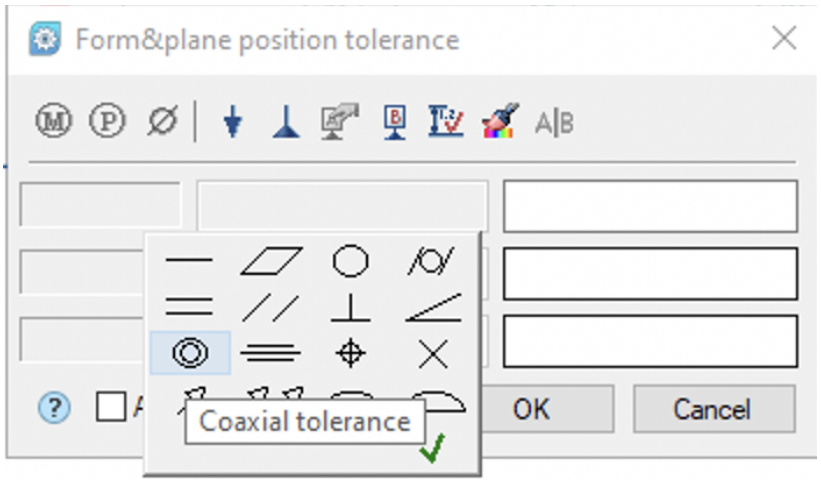


Fig. 45. Choosing tolerance type

Right-click on the second field and select the value of the tolerance deviation 0.01 from the context menu by expanding the table of the standard series (Fig. 46).

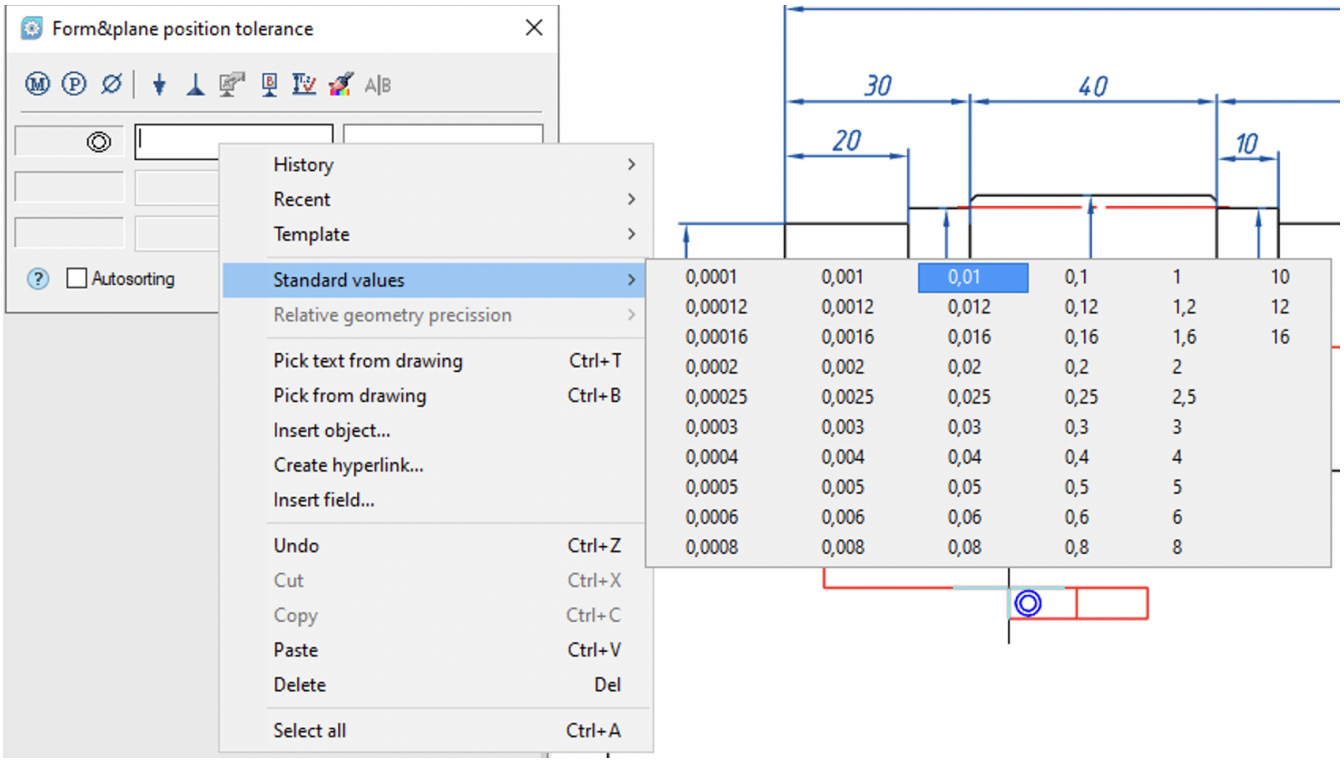


Fig. 46. Selecting a tolerance value from a standard range

Click on the third field and choose ***Specify base*** (Figure 47). Follow the instructions on the command line and select the base ***D*** previously installed in the drawing. The symbol is automatically displayed in the ***Form&plane position tolerance*** window. Click ***OK***.

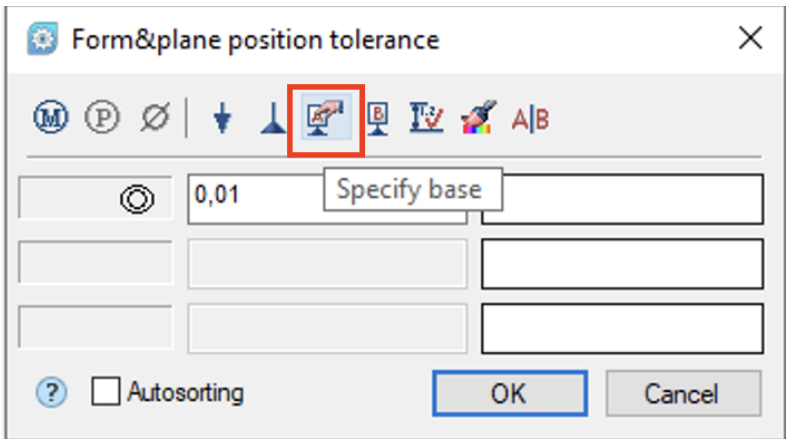


Fig. 47. Base specification

5.3. Install additional leaders to the fourth and fifth sections of the shaft: double-click on the tolerance designation and go to its editing window.

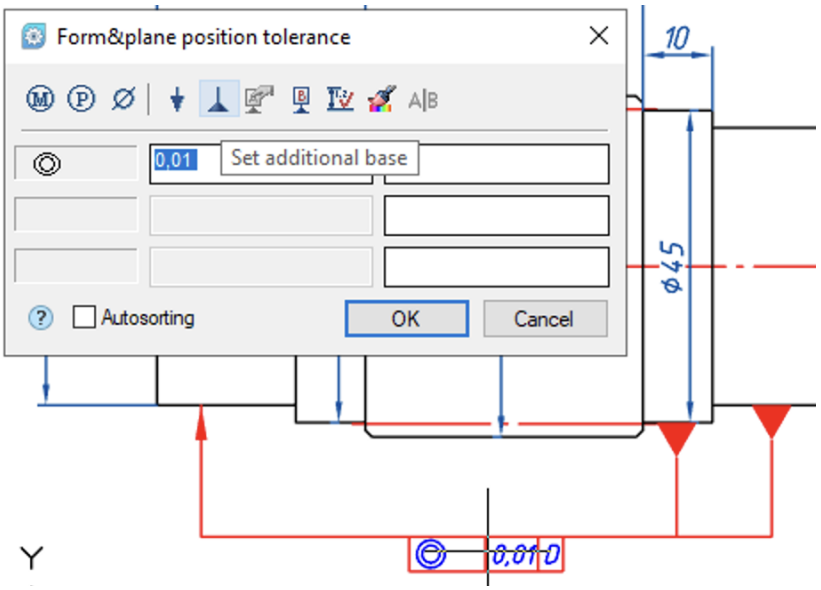


Fig. 48. The command for setting additional leader lines

Click on *Set additional base* command. Specify an object. Bring the leader line to the tolerance mark (Fig. 48).

5.4. Change the display of the leader's end.

Select the leader. Click on the triangular marker that appears on the right end of the leader. In the list appeared select the ***Arrow*** option (Fig. 49). The leader type can also be changed in the context menu: the ***leader lines*** settings. Repeat the procedure for the remaining lines.

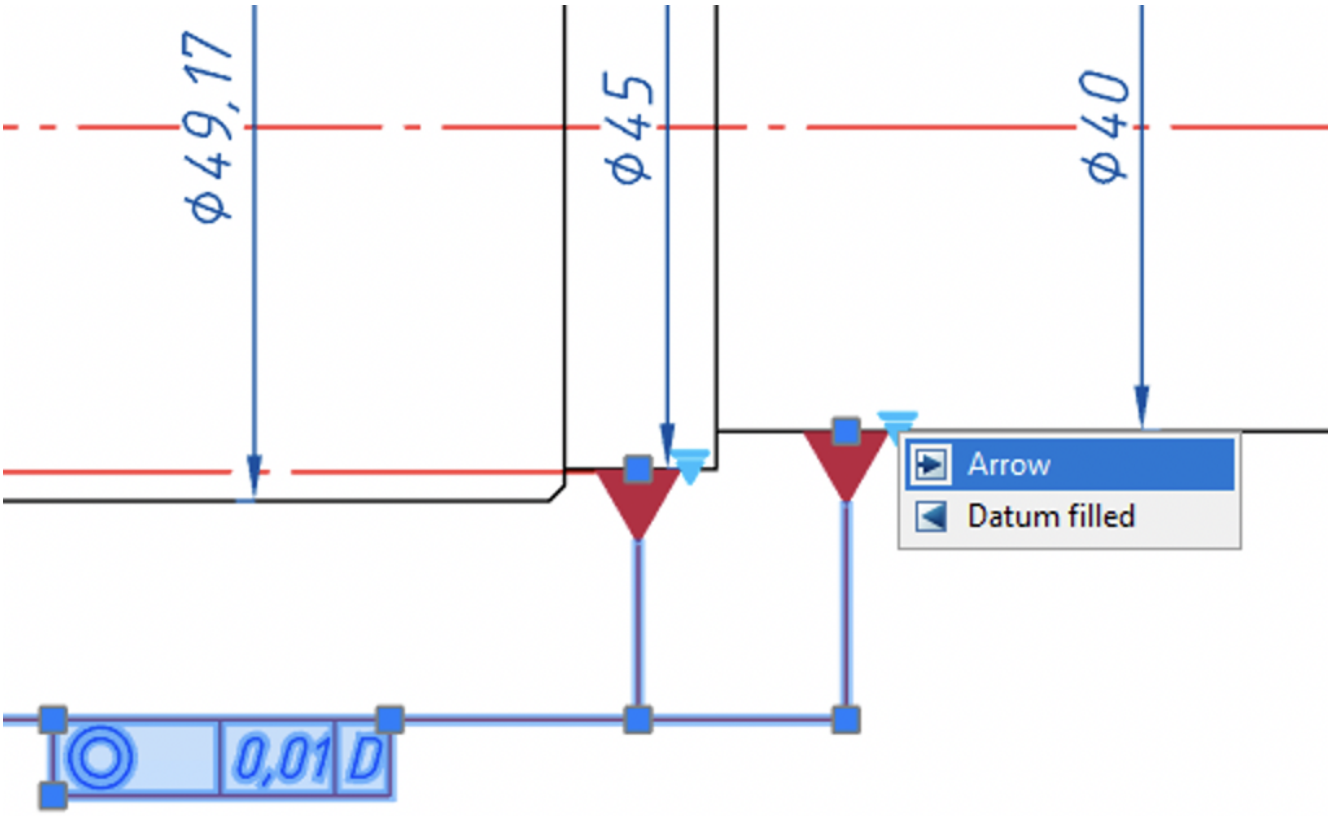


Fig. 49. Changing the arrow type of the leader line.

5.5. Insert a roughness symbol.

Call the ***Surface texture*** command by clicking the corresponding button on the *Symbols* tab or through the path in the classic interface: ***Mechanica*** → ***Symbols*** → ***Surface texture*** → ***Surface texture*** menu.

Move the cursor to the left of the first section of the shaft, up to the extension line (Fig. 50) and right click.

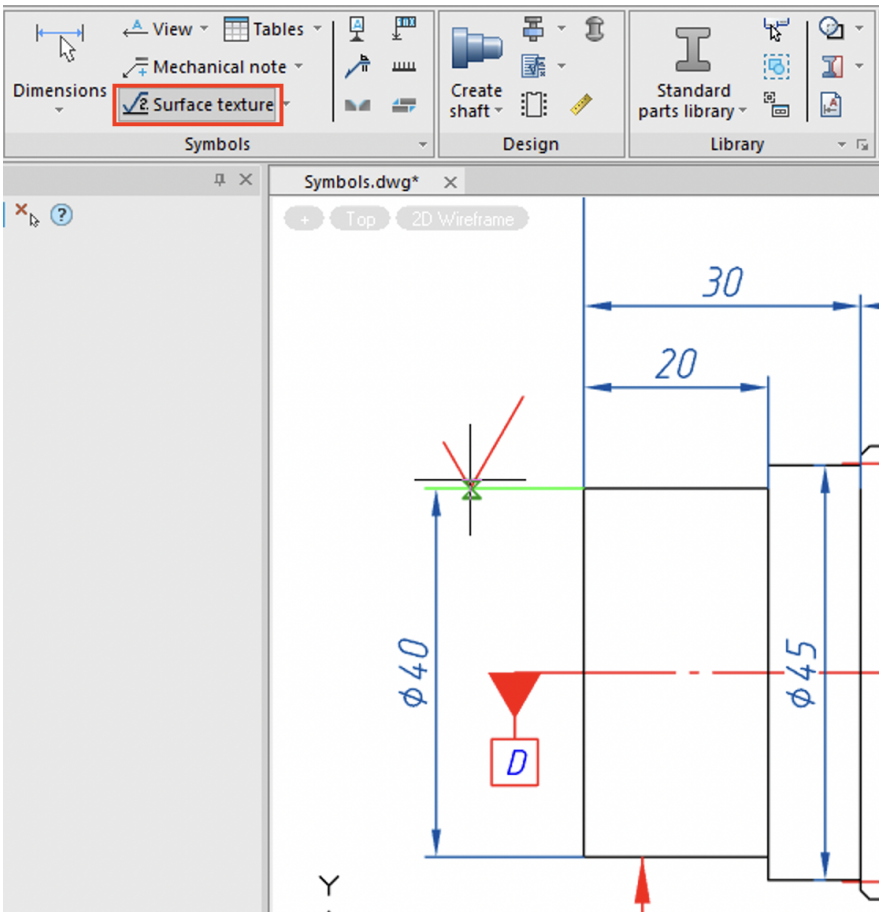


Fig. 50. Surface texture

In the dialog box that appeared right-click to call the context menu of the upper field under the flange and select the roughness value Ra 2.5 (Fig. 51).

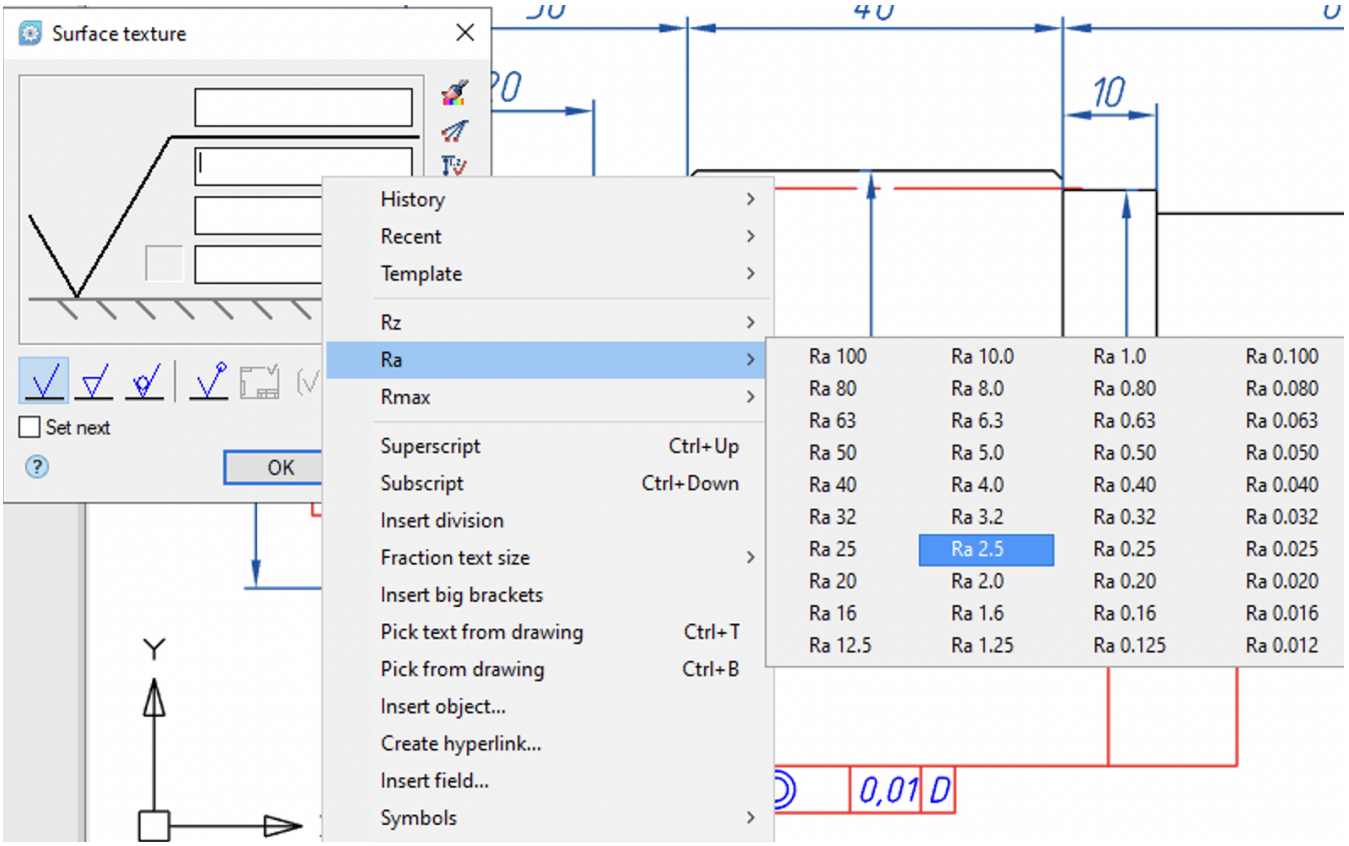


Fig. 51. Selection of the roughness value from the standard Ra list

Activate the ***Set next*** option to save the selected roughness value for the next symbol settings. Click ***OK***. (Fig. 52.)

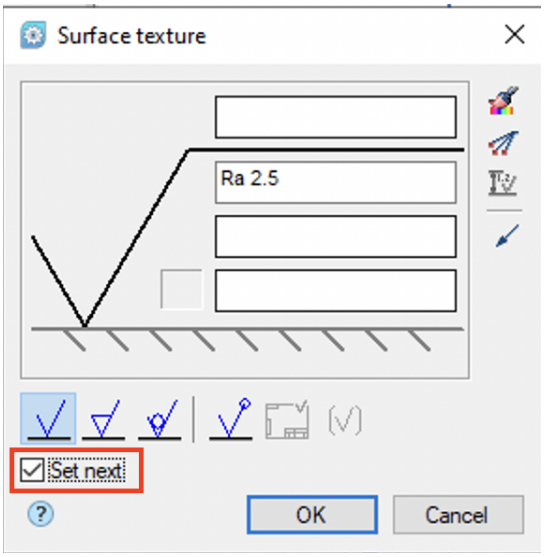


Fig. 52. *Set next* option

5.6. Break symbols

On the ***Utilities*** tab, call the ***Break lines*** command . Specify the first point of the break line on the fifth section of the shaft and move the mouse cursor down. Call the context menu and select ***Two curved*** (or ***Mechanica*** → ***Utilities*** → ***Break lines*** → ***Two curved*** menu). Click at the bottom point of the break (Fig. 53), move the mouse cursor to the right and enter the break width - 2.

Press ***Enter***.

Set a break on the first section of the shaft.

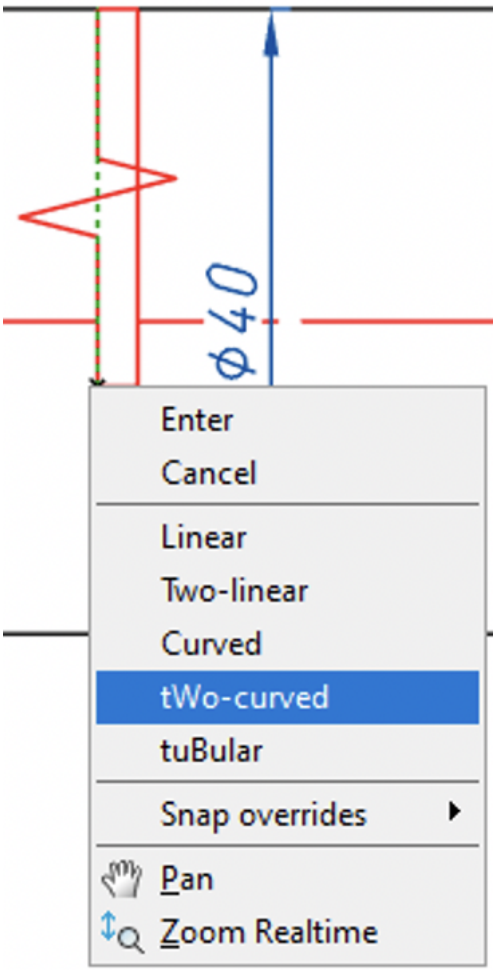


Fig. 53. Break symbols

Edit the values of the dimensions of the shaft sections with breaks and define them as 50, 60 and 100. Set the total length as 310.

The final view of the shaft with installed breaks is shown in Fig. 54.

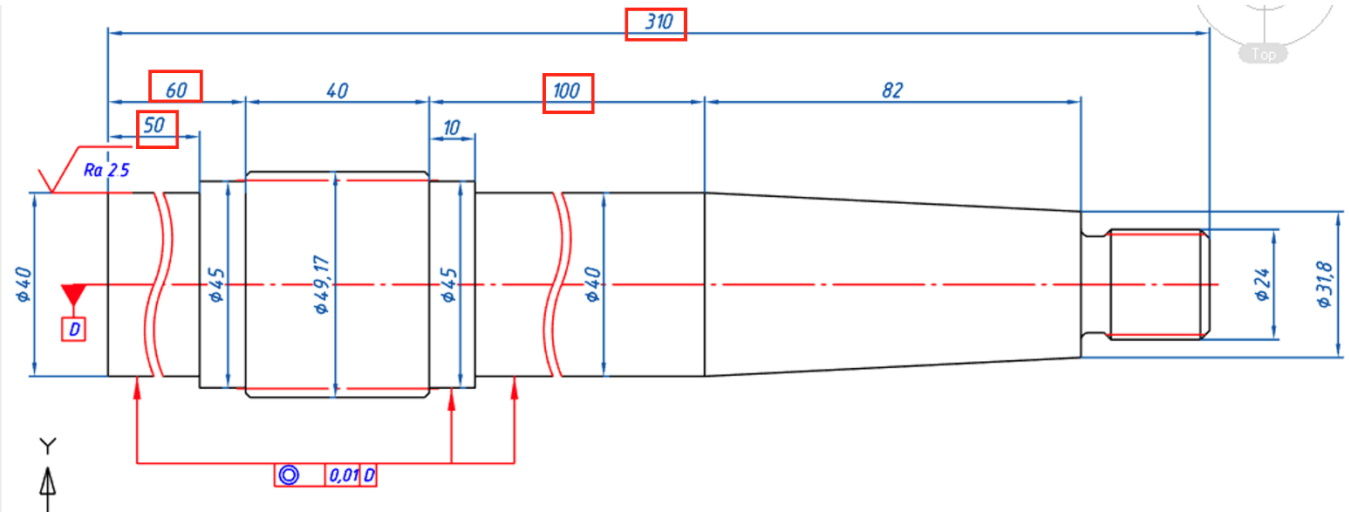


Fig. 54. Shaft with breaks

5.7. Setting the dimension tolerance.

Double-click on the diameter 40 of the fifth shaft section to open the edit window. Call the method of recording tolerances and replace it with a letter designation (Fig. 55).

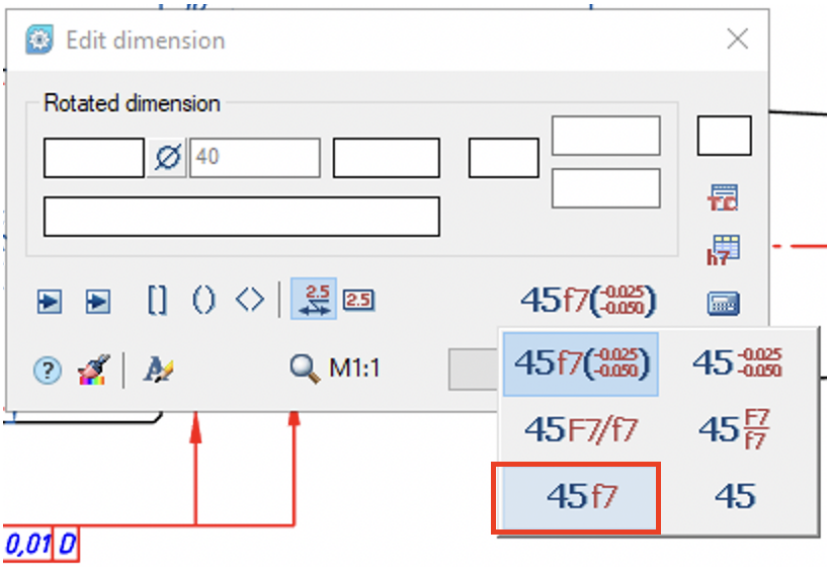


Fig. 55. Dimension editing window

Set the tolerance: open the dimension editing and press the ***Fit table*** button . In the ***Tolerances*** window switch to the ***Shaft*** tab and set the tolerance value as k6. Click ***OK*** (Fig. 56).

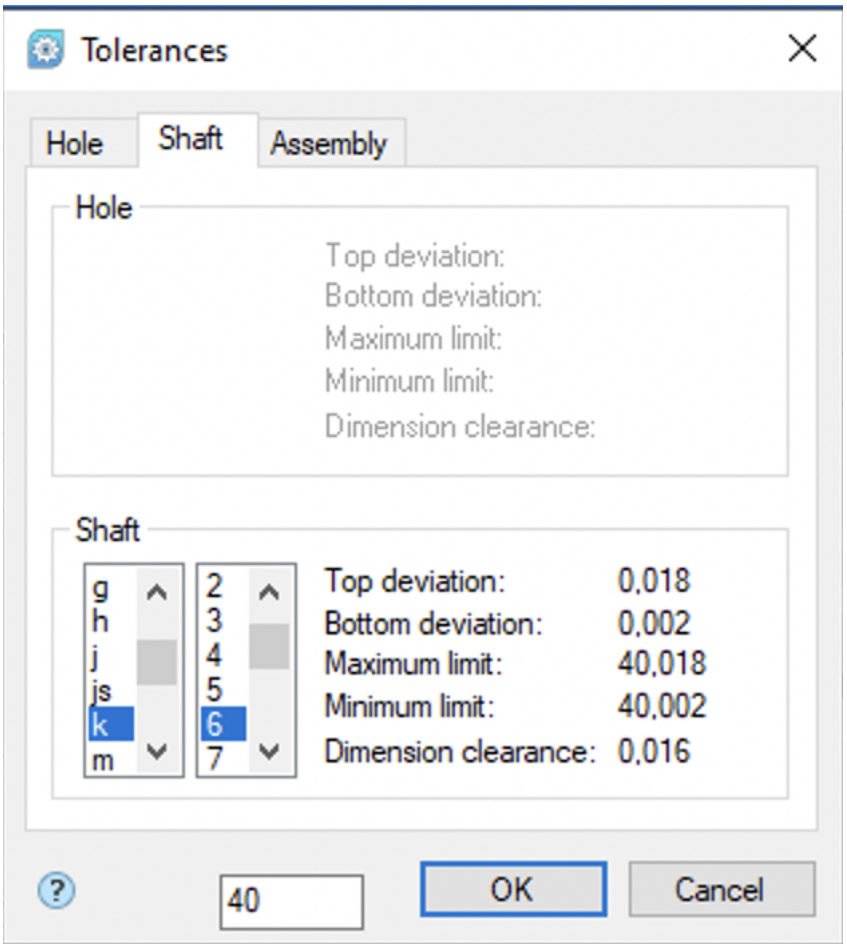


Fig. 56. *Tolerances* dialog

Repeat for the fourth shaft section.

5.8. Copying properties.

Proceed to edit dimension 40 of the first shaft section.

Click the ***Match Properties*** button  in the dimension edit window. Select the previously edited 40k6 size.

The properties are automatically set as for the specified first section size: tolerance k6.
Confirm the action by clicking ***OK***.

The final view of the shaft with the referenced base, roughness, gaps, dimensions and tolerances is shown in Fig. 57

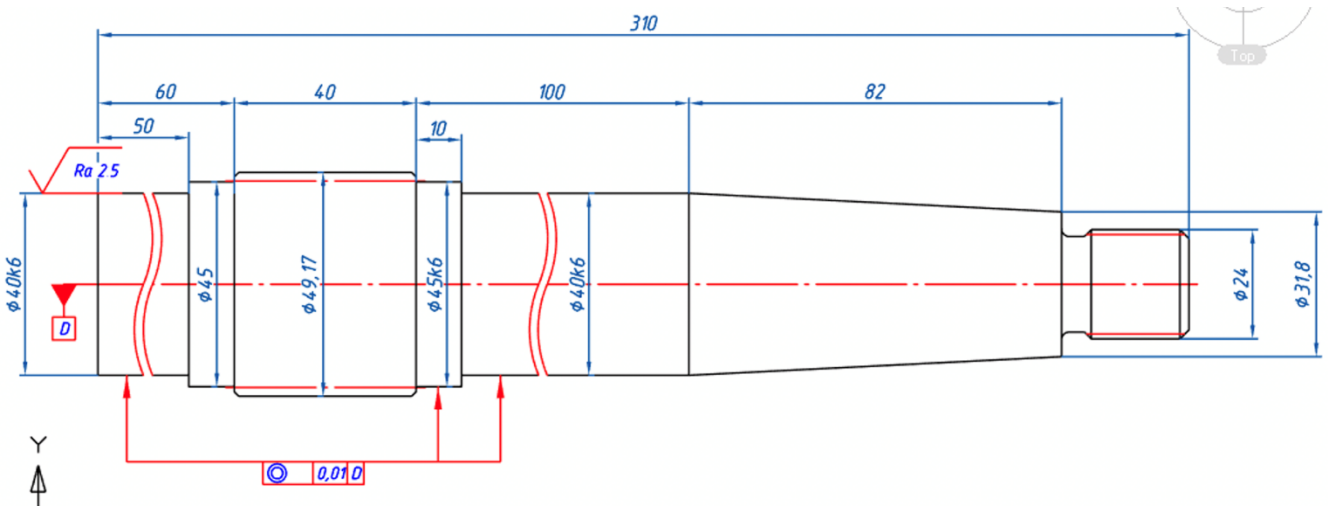


Fig. 57. Shaft with edited design

Save the file to the ***Exercises*** folder.

6. Installation of Fasteners

Installation of fasteners in nanoCAD Mechanica can be done by dragging and dropping from the library, or by using the ***Threaded fastening*** function.

Installation of fasteners by dragging from the library is recommended when the fasteners are a single product, not a package (set of components). The ***Threaded fastening*** function makes it easier and faster to install fasteners that consist of several standard items.

6.1. A single fastener installation with the example of rivets.

Open the ***Installation of rivets.dwg*** file in nanoCAD Mechanica located in the ***Test drive nanoCAD Mechanica/Installation of fasteners*** folder.

Open the dialog ***Library*** → ***Standart parts library*** or use the following path in the classic interface: ***Mechanica*** → ***Standart Parts*** → ***Standart parts library***.

Expand ***Fasteners*** → ***General machinery*** → ***Rivets*** → ***Extended precision*** and select the flat-oval head rivet GOST 14800-85 (Fig. 58).

Move the selected element to the upper line of the workpiece 1. Once the snap is triggered, click to fix the rivet insertion point.

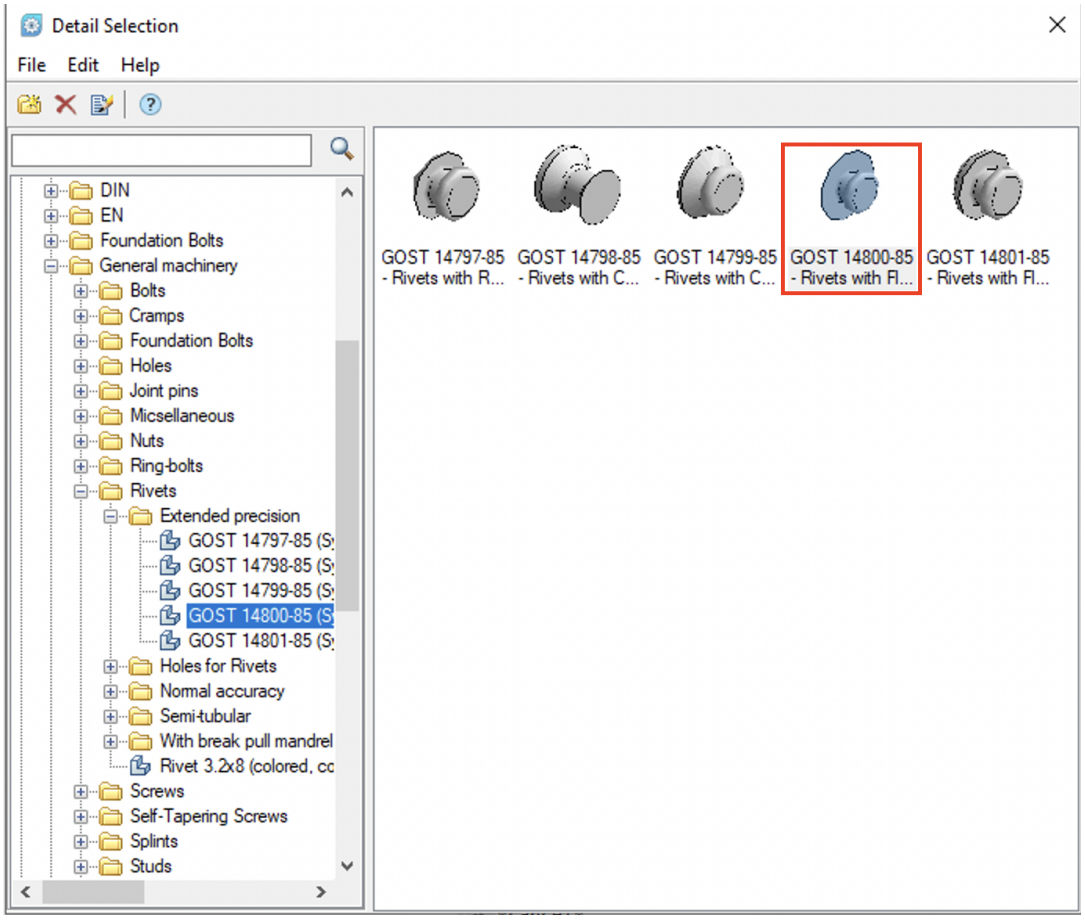


Fig. 58. Selecting and inserting a rivet

Turn on ***ORTHO*** mode (F8) and specify the insertion direction.

In the new window, click the ***Measure distance*** button (Fig. 59) and specify the upper and lower points of the plates to be fastened.

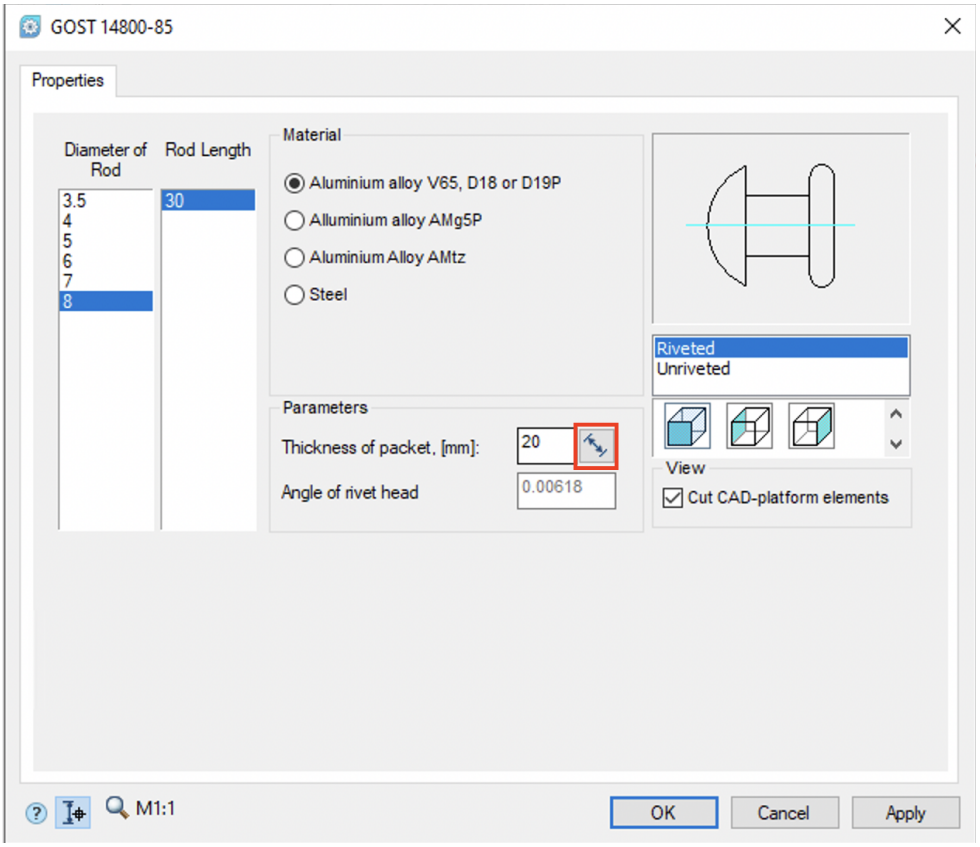


Fig. 59. Distance parameters of the rivet

The ***Thickness of packet*** will show the corresponding value. The rivet shank parameters will change accordingly.

Select the ***Unriveted*** rivet option and click ***OK*** (Fig. 60)

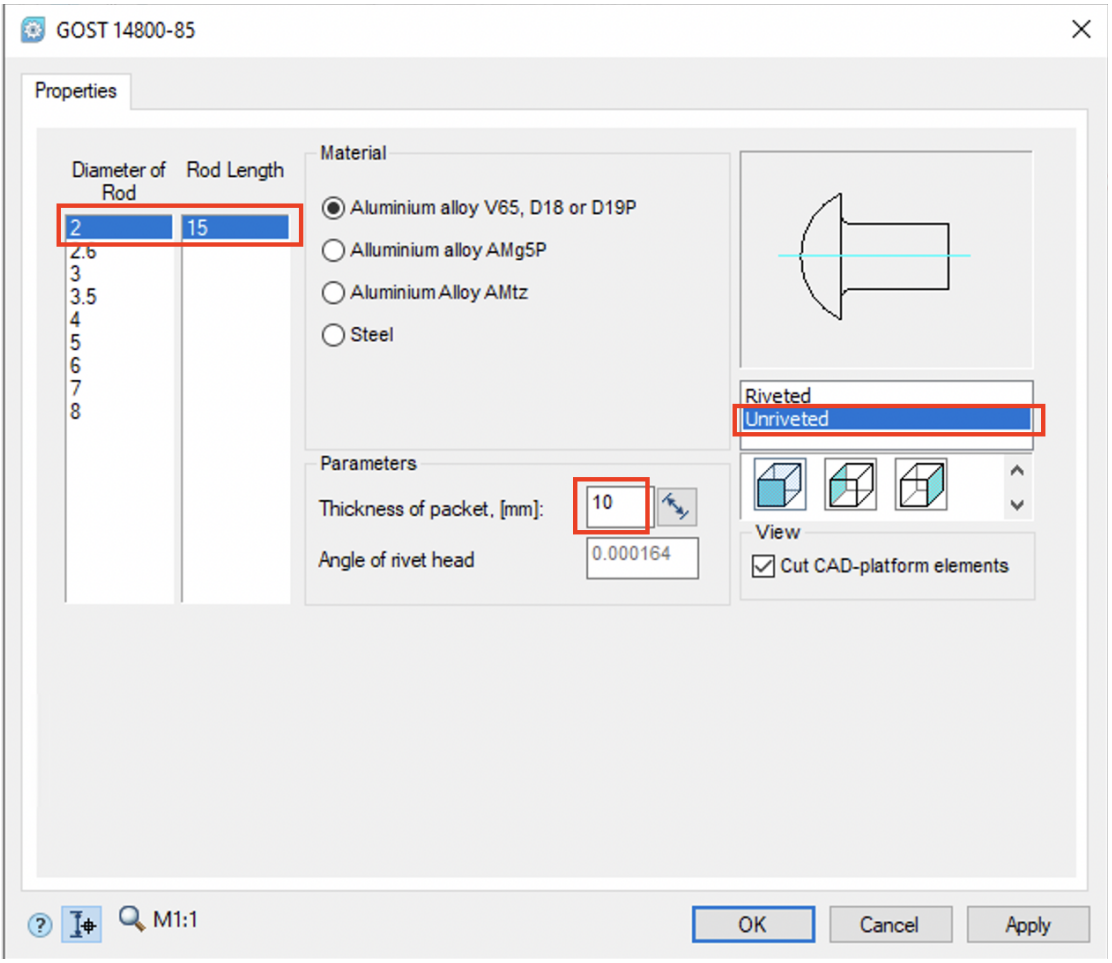


Fig. 60. Changed rivet parameters

Install the ***Riveted*** rivet in the same way. No need for the thickness of the packet to be measured, since the system remembers the previous value.
Install the ***Rivet with countersunk head GOST 14798-85*** on the second package.
The final version of the installation of rivets is shown in Fig. 61. Save the file to the ***Exercises*** folder.

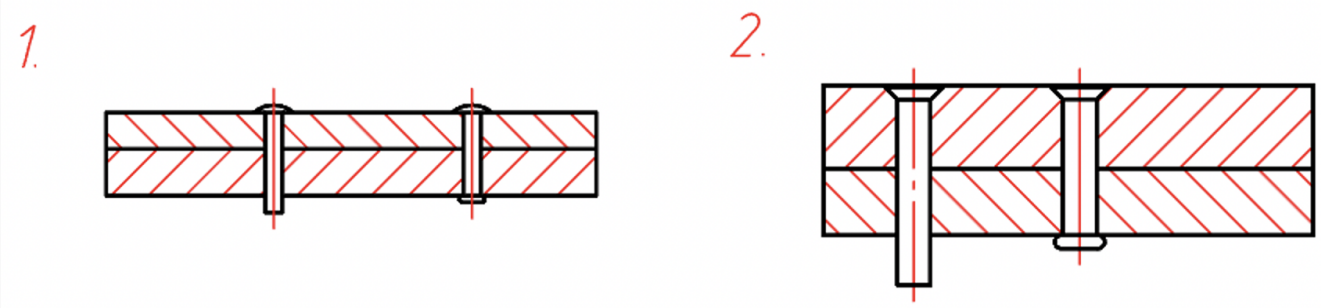


Fig. 61 Final view of installed rivets

6.2. Let's install the fastening by using the ***Threaded fastening*** function.

Open the ***Installation of bolted and screw connections.dwg*** file in nanoCAD Mechanica, located in the folder ***nanoCAD Mechanica Module Test drive / Installation of fasteners***. Click on the ***Threaded fastening*** button on the ***Design*** tab (Fig. 62).

Fig. 62. The ***Threaded fastening*** command

Move the cursor to the top line of the first sketch. When the line is highlighted in green, click on it. Fix the position of the attachment in the ***ORTHO*** mode with another click outside the package (Fig. 63).

The ***Fasteners*** dialog box will appear. By default, some mounting options will be offered. You can visually evaluate it on the *Preview* tab and adjust or replace the products and structural elements of the package on the ***Template*** tab.

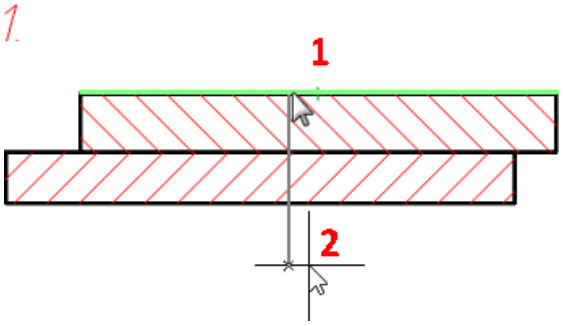


Fig. 63. Selecting the location of the bolted connection

Click on next to ***Screw*** in the box. The part selection dialog will open.
Find (you can use the search) and select ***GOST 7783-81 - Cup nibbed head bolt***. The information on the ***Template*** tab is updated.

Next, select the washer, nut and split pin in the same way:

- washer GOST 11371-78
- nut GOST 5933-73
- split pin GOST 397-79

The final view of the modified template is shown in Fig. 64.

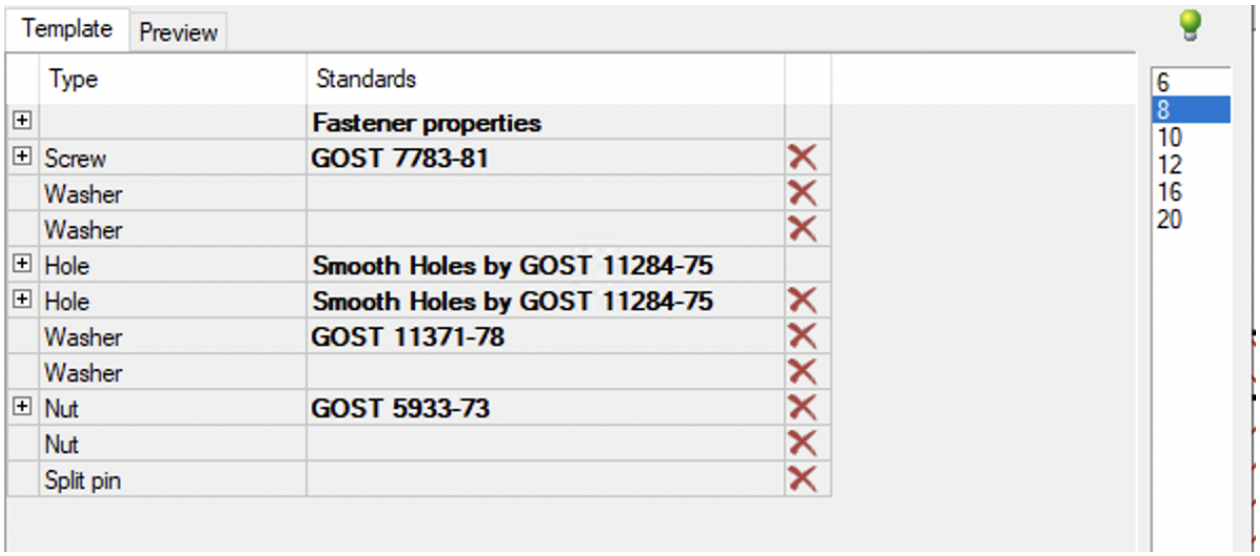


Fig. 64. Changed template

Click ***OK***. The fastener from standard products selected through the dialogs of the ***Threaded fastening*** function will appear in the previously indicated place of the sketch (Fig. 65).
Adjust the holes built with the bolted connection if necessary: double-click on the centerline to enter the fixture editing mode.

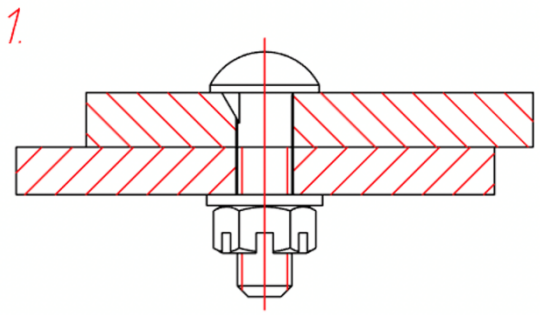


Fig. 65. The final view of the bolted connection

By using this option, the new fastener package will not be saved in the list of templates in the ***Fasteners*** dialog.

In the second sketch, construct the screw connection by creating a new template.
In order to create a new template in the ***Fasteners*** dialog box, click on the corresponding button located in the upper left corner (Fig. 66). Enter the new template's name.

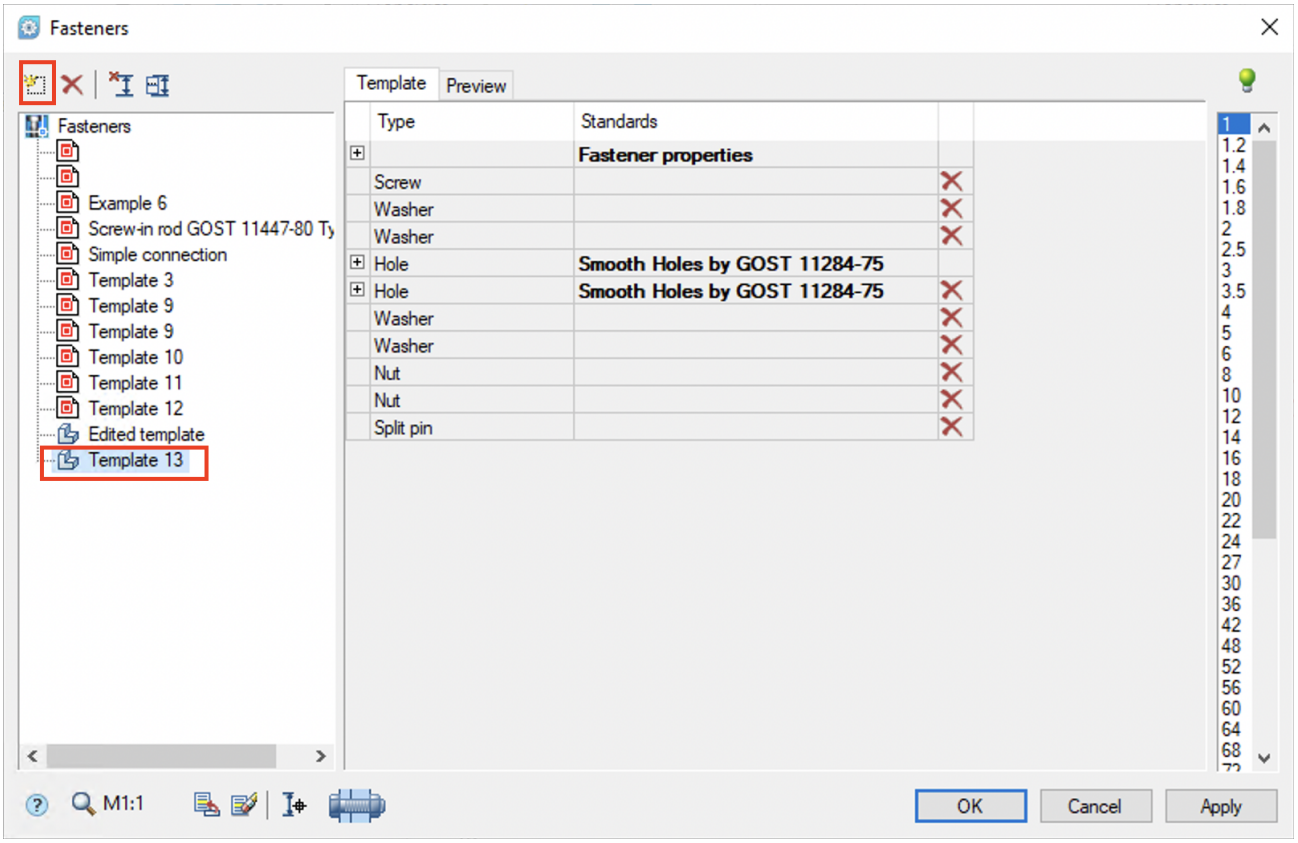


Fig. 66. Creating new template

Add a GOST 17473-80 screw, and then add a washer and holes: smooth according to GOST 11284-75 and thread blind.

Correct the screw length and edit the hole parameters (fig. 67).

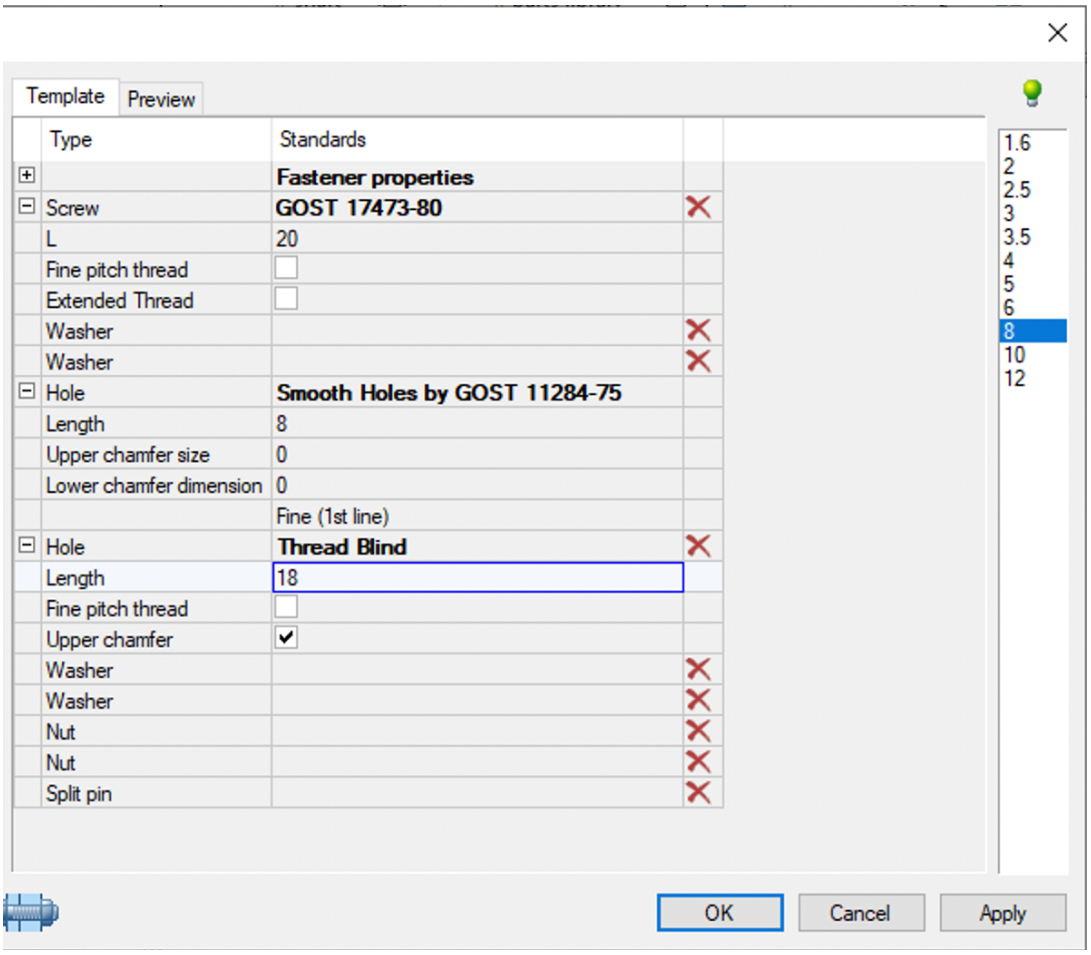


Fig. 67. Adjustment of connection package parameters

The final view of bolted and screwed connections is shown in Fig. 68.

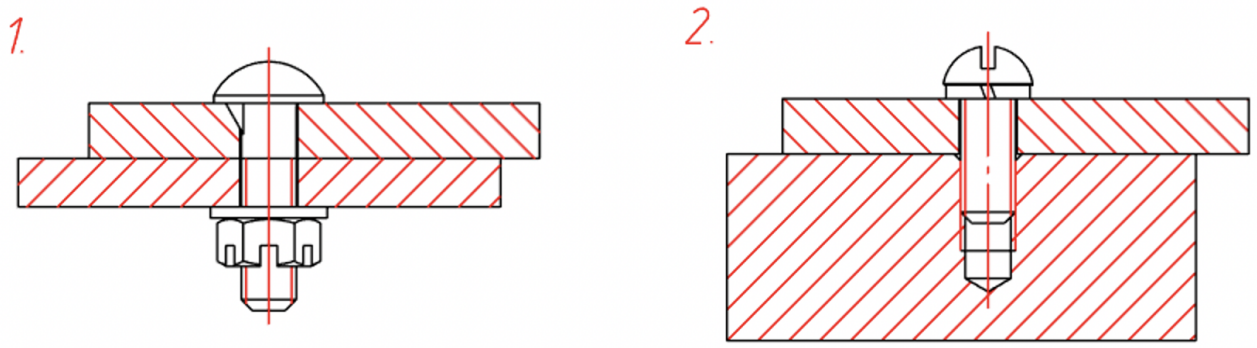


Fig. 68. Final view of bolted and screw connections

Save the file to the ***Exercises*** folder.