CAD/CAM System N-Ship+

Version 6.0

Module MDET

Formation of ship case designs and elements

NSHIP.00006.006-2025

User manual

Sheets 104

Saint Petersburg

2025

ANNOTATION

This document is a user's guide for working with MDET module, which is a part of N-Ship+ automation system and functions in nanoCAD graphic processor environment.

The module is designed to perform detailing of hull structures, production of technological documentation and preparation of documents on special technological (bending, assembly) tooling.

The manual includes the description of the module settings, the methodology of dialogue work of the user with hull models, formation of details of ship structures, service procedures and utilities, features of interaction with other modules of the system.

The document is intended for specialists who operate N-Ship+.system for design and technological preparation of hull production and have practical experience of working with nanoCAD.
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СОДЕРЖАНИЕ

GENERAL INFORMATION

1.1. Purpose of module

The MDET module is designed for working with three-dimensional hull models of ship structures and, on its basis, for forming sheet and profile parts of the ship hull and their processing technology. It contains a wide range of service functions and provides integrated communication with
N-Ship+ modules.

After installation of the N-Ship+ system software, the MDET module is ready for use.

1.2. Terms and agreements

The expression ‘select’ an object used in the document means the requirement to select an object on the screen using the keyboard or mouse.

Keyboard selection: position the cursor on the object using the arrow keys and press the Enter key.

Selecting with the mouse: position the mouse pointer on the object and click the left mouse button.

The following font conventions are used throughout this manual:

italicized- directory names, file names and file types, explanatory text to graphical editor queries in module commands;

bold font - names of modules and components of the software complex of the system

N-Ship+, menu items, names of buttons and keys, names of commands of the software complex in the text of the dialogue with the graphical editor nanoCAD;

all uppercase - names of layers, commands and names of named objects.

1.3. Module interface

The MDETl module requires the MDET drop-down menu, which is automatically loaded using the NSHIP\Sys\nship64.package file, if it is specified in the Autoload window called from the Load/Unload Applications dialogue box (Figure 1).

In the menu bar, a two-character suffix is added to the end of the menu title to define the language of the current localisation: MDETen (English) (Figure 2). For the sake of brevity, the menu will be referred to without the suffix (MDET) later in the document.

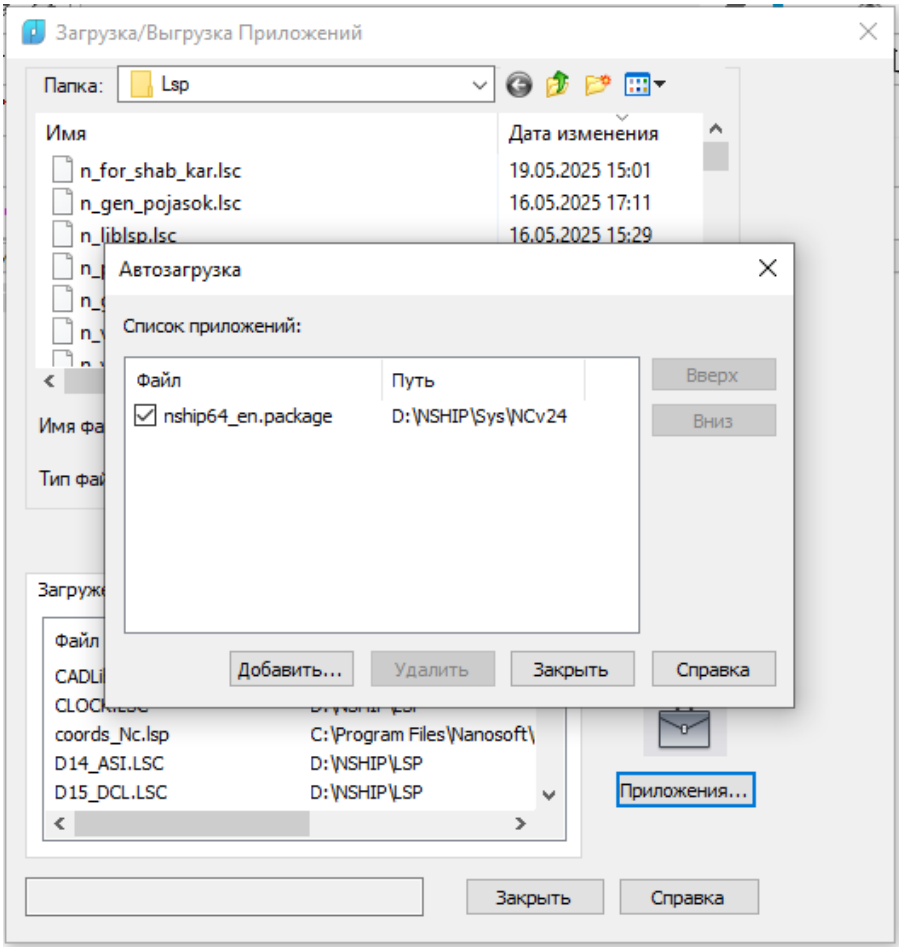


Figure 1. Loading nship64.package

Figure 2. System menu headings **N-Ship+**

The MDET module provides the user with a set of commands (functions), which are primarily launched using the MDET drop-down menu (Figure 3).

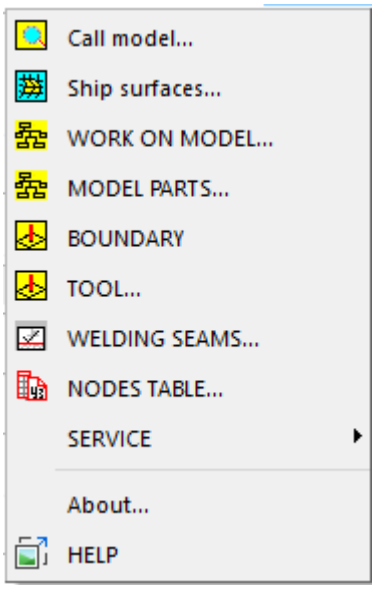


Figure 3. Dropdown menu **MDET**

The full set of functions of the MDET module is also available through the MDET toolbar (Figure 4).

Figure 4. Toolbar **MDET**

All commands of the module use the context menu (right-click menu).

All dialogue windows of the MDET module are provided with tooltips when holding the mouse over the corresponding internal window.

When the mouse is within a dialogue box, pressing the F1 key starts the MDET module help procedure.

Attention: All windows include a minimise/unmodify button for ease of use.

2. WORK IN THE MODULE ENVIRONMENT

2.1. Starting work with the module

Before using the module, it is necessary to perform installation on a specific project for the active user using the main menu command **BDATAen->ORDER**.

2.2. Main menu of the module

Figure 3 shows the main menu of the MDET module.

The call **Call model…** command of the MDET module allows you to operate with various models (individual dwg files) linked to the drawings of the active project.

The basic principle that is implemented in the MDET module for linking dwg files and drawings is as follows:

- all information about the project is concentrated in the project database;
- information in the database is formed according to drawings;
- drawing primitives can be placed in one or more dwg files.

Command **Ship surfaces…** allows to form multifaceted networks representing ship surfaces in the model:

- by the prototype (file with the extension .fef) created in the modelling program FREE!ship (GNU GPL);
- indicating the points of the frame;
- on a set of lines;
- by extruding the edge of the existing surface;
- by default, using the parameters of the current project.

Command **WORK ON MODEL…** allows you to generate various objects in the model, from a point to surfaces, as well as to find points and lines of intersection of objects with each other.

Command **MODEL PARTS…** allows to form sheet parts on flat sections of the model, reamer and bending frame for non-planar parts of the model, profile flat and bent parts, bookends and belts.

Command **BOUNDARY** allows to identify a flat closed contour on the model cross-section.

Command **TOOL…** allows you to create the assembly tooling for a section in the active model.

Command **WELDING SEAMS…** allows you to create a database of project welds and generate chamfer parameters on the edges of the part.

Command **MODES TABLE…** allows to form a table of structural units in the project database

Commands of submenu **SERVICE** contain service commands of the module **MDET**.

Command **About…** calls the window with module details to the screen **MDET**.

Command **Help** calls the module's help system **MDET**.

3. MODELLING

3.1. Command window interface **Call model…**

Figure 5 shows the function panel called by the command **Call model …**

The panel contains the tree of objects of the active order.

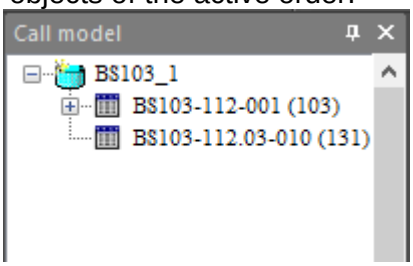


Figure 5. Functional panel of the command **Call model …**

Right-clicking the cursor in the object tree opens a context menu, the contents of which depend on the type of object clicked.

The context menu of the <Project_Part> object is shown in Figure 6.

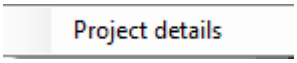


Figure 6. Context menu of the drawing object

If you click on Project details, a window with details of the active project is shown in Figure 7.

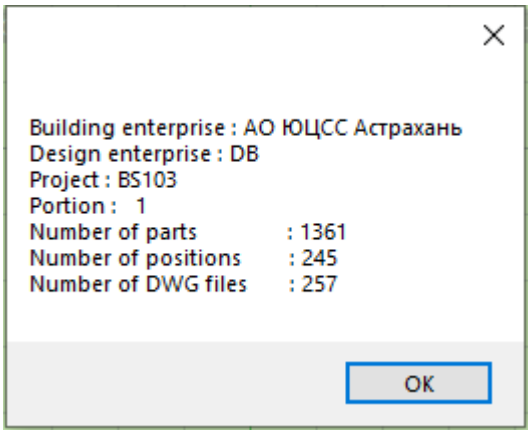


Figure 7. Project details

Attention: The <Project_Part> tree shows the order in progress and registered for the active user in the BDATA module.

The context menu of the <drawing> object is shown in Figure 8.

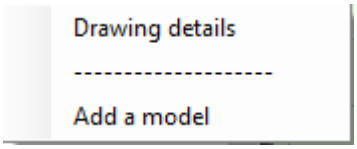


Figure 8. Context menu of the drawing object

If you select Drawing details, a window with drawing details is shown in Figure 9.

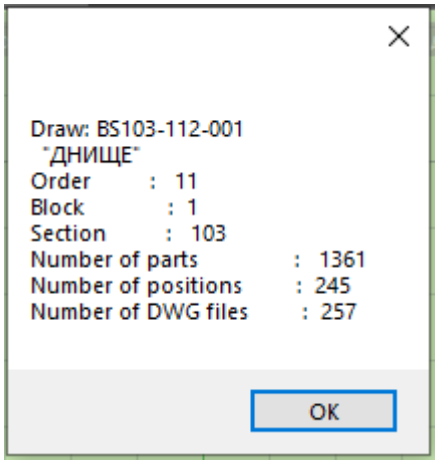


Figure 9. Drawing details

When you select Add a model, the window shown in Figure 10

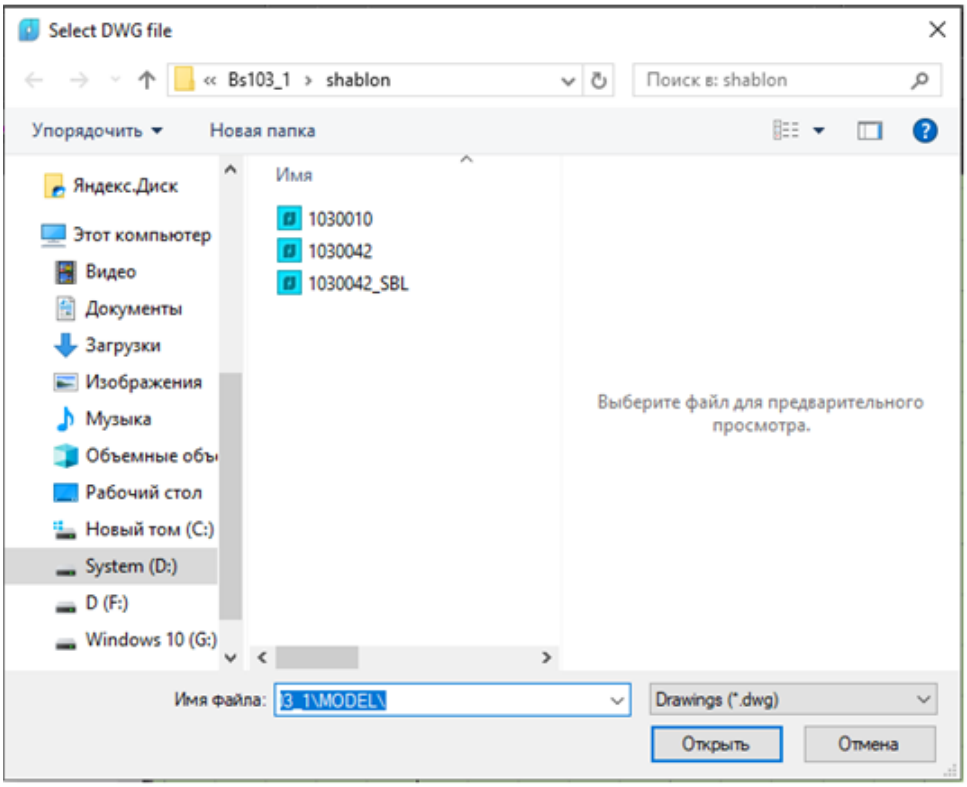


Figure 10. Model selection dialogue box

In this window it is necessary to set the access path, select the model file and click the Open button.

Attention: In case of selecting a model file not from the <Model> catalogue of the current project and if there is such a file in the <Model> catalogue, a request to copy the model file appears.

As a result, the name of the model file will appear in the tree under the specified draw as shown in Figure 11.

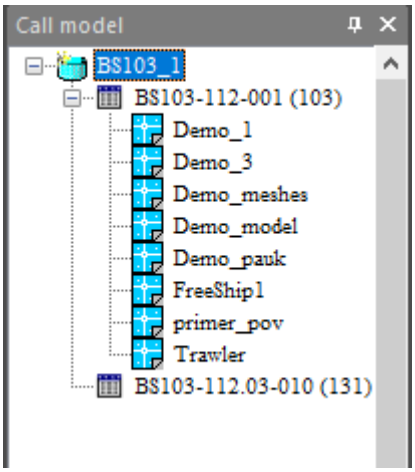


Figure 11. Tree model under the specified draw

The context menu of the <model> object is shown in Figure 12.

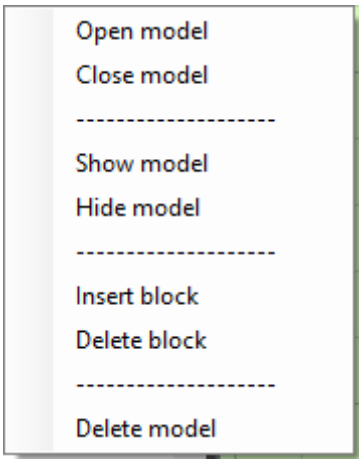


Figure 12. Context menu of the model

When you select Open model in the tree, the model file is loaded into the nanoCAD workspace and the prompt shown in Figure 13 pops up.

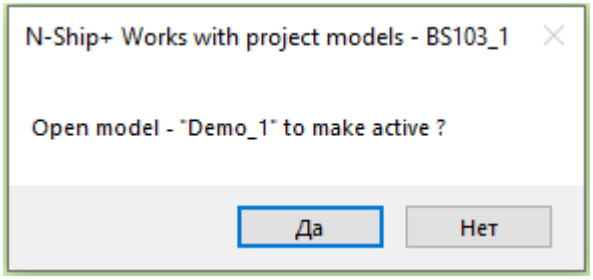


Figure 13. Context menu of the model

Attention: When working with a drawing, only one model can be active, i.e. the one in which the model is being worked on. After opening the model file, if you answer affirmatively to the request to make the model active, this model becomes active and the icon in the drawing model tree is coloured red.

If you select Close model, the window shown in Figure 14 will pop up.

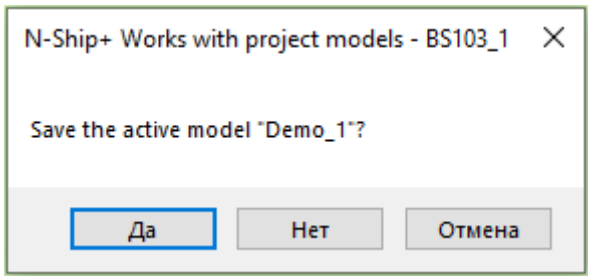


Figure 14. Dialogue window requesting to save the model

If you press the Yes button, the model file will be written, and if you press the No button, the model file will not be written to the database. If you click on the Cancel button, everything remains in the same state.

For an active model, selecting the context menu item Show model causes the procedure of highlighting the model primitives, and selecting the item Remove model causes the procedure of cancelling the highlighting of the model primitives.

It may be necessary to have primitives of other models in the editing session, either to copy them or to bind to them. Such models can be inserted into the active model as disjoint blocks.

If you select the Insert block item, the primitives of the model specified in the tree will be loaded into the active model, and the icon of the model in the tree will be green.

If you select Delete block, the window shown in Figure 15 will pop up.

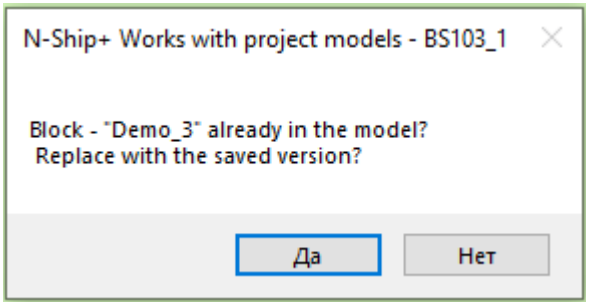


Figure 15. Dialogue window with a request to delete a block

If you press the Yes button, the primitives of the block specified in the tree, the icon of which is green, will be unloaded from the active model.

Attention: Before deleting a block, if it is necessary to keep the primitives of the block in the active model, it is necessary to copy using nanoCAD tools.

When selecting the Delete model item, the window shown in Figure 16 will pop up.

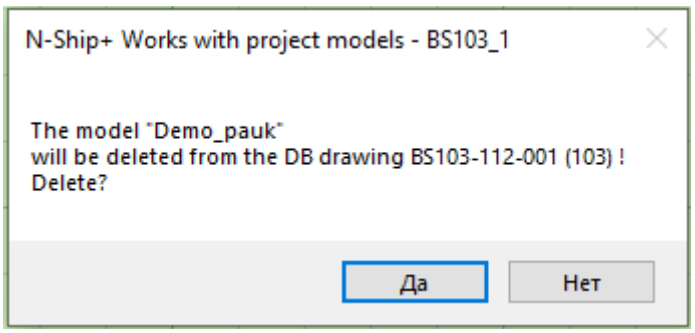


Figure 16. Deletion request dialogue box

If you click on the Yes button, the procedure of deleting a model from the DB drawing models list is called. The model file is not deleted from the project model catalogue.

4. FORMATION OF SHIP SURFACES

4.1 The window interface of the Ship surfaces command...

Formation of ship surfaces is possible only in the model, which must be opened with the command **Ship surfaces ….** The model must be initialised using the Model module of the N-Ship+ system, otherwise the window shown in Figure 17 will pop up when the command is called.

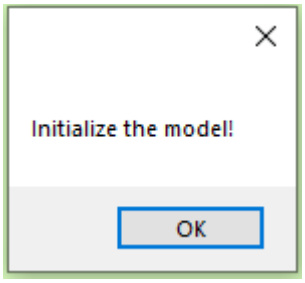


Figure 17. Request for model initialisation

Figure 18 shows the function panel invoked by the **Ship's surfaces …**

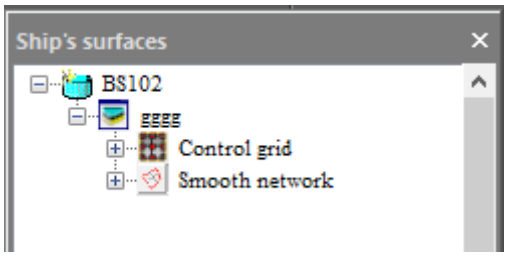


Figure 18. Function panel Ship's surfaces

The panel contains a tree of model surfaces, where the top node is the project name.

The context menu of the <Project> object is shown in Figure 19.

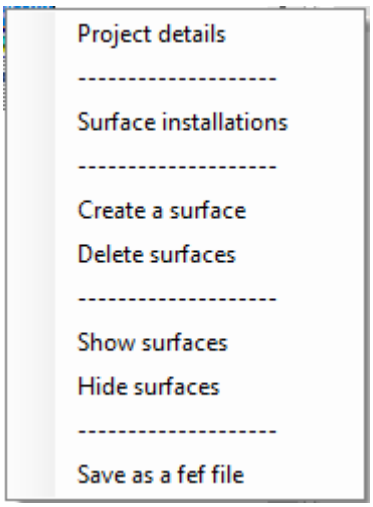


Figure 19. Context menu of the model object

Selecting **Project details** item brings up a window with project details.

Selecting the **Surface installations** item brings up the window shown in Figure 20.

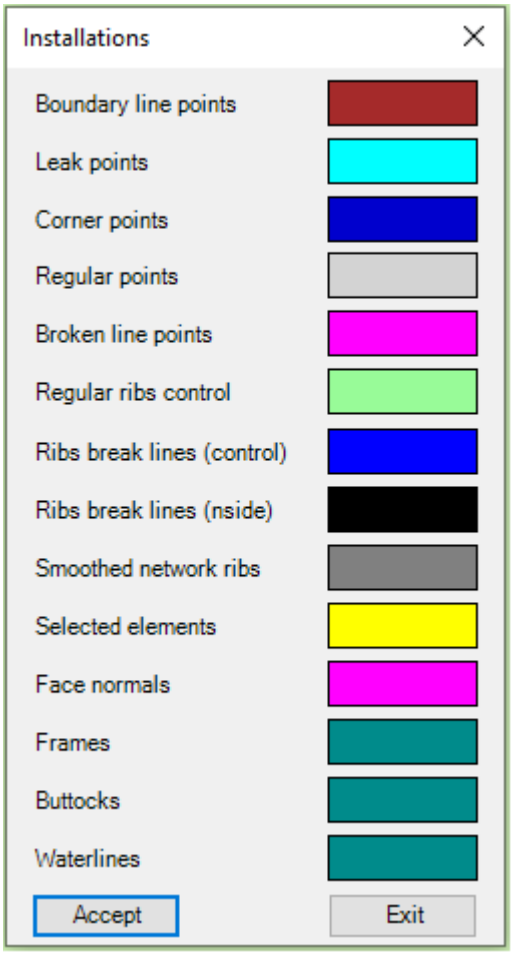


Figure 20 Settings dialogue box

Ship surface formed by the command **Surface installaitions** represents a SubDMesh with faces, edges and points (vertices).

An edge is a surface fragment usually defined by 4 points. Sometimes a face can have 3 or more than 4 points, but the best results are obtained when most faces consist of 4 points.

All points of a face are connected by two types of edges - boundary and regular.

A boundary edge is characterised by the fact that it always has only one edge associated with it. Examples of boundary edges are the upper deck contour line (UD) or the diametral plane line (DP).

Regular edges are always far from the surface boundary and must always be separated by two adjacent faces. Two faces separated by an edge are joined smoothly along their public edge. It is possible to specify an edge as an edge of a break line, then the faces are connected at an angle to each other.

The points form the base of the surface. Editing the surface is based on moving, adding or deleting points. Points are of two types - regular and angular.

Regular points are all points except corner points, these points have some offset from the surface. This offset from the surface is greater in areas where the curvature is greater and becomes smaller the more points and edges are inserted.

Corner points are defined points, usually associated with two or more break lines. Just as a break line edge can be used to determine that two faces should be at an angle to each other, corner points can be used to do this with two adjacent edges. Corner points are the only type of points actually located on the surface.

The window shown in Figure 16 is used to select the colour of the corresponding surface objects.

The colour of the buttons can be changed by clicking on the button and selecting the colour in the **Colour** window that pops up.

If you click on the **Reset** button, the color of the buttons will be set by default.

If you press the **Accept** button, the color of objects in the model will be set according to the color of the corresponding buttons.

The **Create a surface**, **Delete surfaces**, **Show surfaces** and **Hide surfaces** items (Figure 19) are intended for creating or deleting project surfaces, as well as for visualizing surfaces on the screen.

Selecting the **Create a surface** item opens the window shown in Figure 21.

The upper part of the window contains the project parameters, and the lower part contains the switches and the **Apply** and **Exit** buttons.

Attention: The Surface extrusion switch is locked. It will be unlocked if any surface is activated.

The Surface by prototype (.fef) switch is activated and the Apply button is unlocked.

Attention: Files with the extension .fef are generated in the environment of the GNU-licensed software product FREE!ship. In the environment of this program it is possible to form a ship surface with specified parameters.

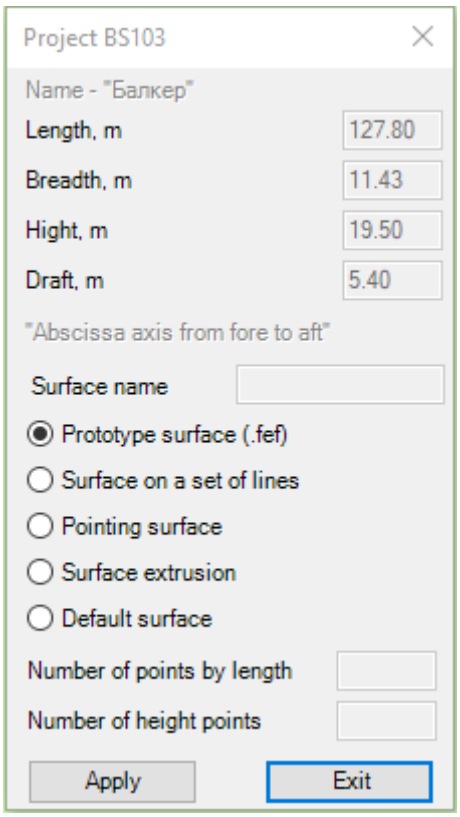


Figure 21. Project dialog box

When you click on the **Apply** button, the window shown in Figure 22 will pop up.

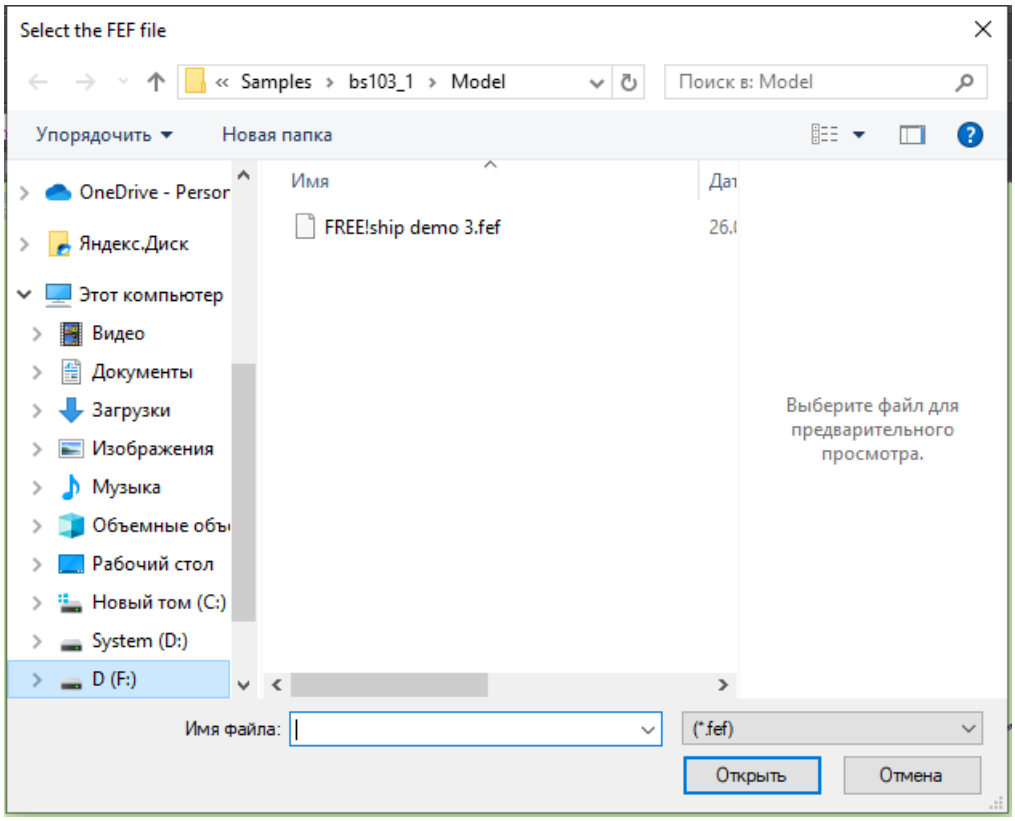


Figure 22. Model call dialog box

In this window it is necessary to specify the prototype file with the extension .fef and click the **Open** button, the model file is loaded into the nanoCAD workspace, and in the functional panel the prototype surface tree is built, which has the form shown in Figure 23.

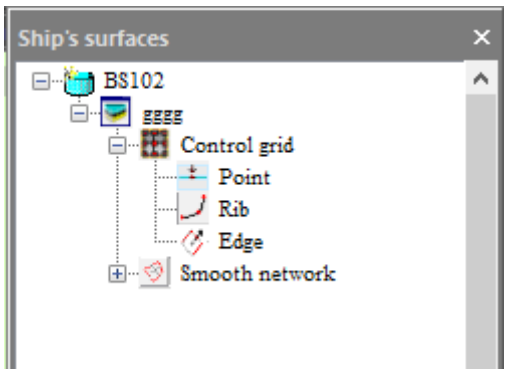


Figure 23. Function panel Ship's surfaces

Attention: When loading the file, the prototype data is scaled to ensure that the model corresponds to the project parameters. The names of surfaces are taken from the prototype file.

Each surface in the model is represented by a Control Grid and a Smoothed Grid.

Attention: All methods of surface formation described below require entering the surface name in the corresponding field (Figure 21), otherwise the **Apply** button will be blocked. The surface name is used to form a layer and an object in the surface tree, so you should strive for brevity. After entering the name in the input field, you must press Enter.

If you activate the **Surface** by line set switch, you must specify in the **Number** of points by length field the number of points of line division by length. When you press the **Apply** button, the procedure of surface formation by the specified sequence of form-forming lines of one direction (all lines must be splines) is started.

When the **Surface** by points switch is activated, it is necessary to set the corresponding values in the input fields **Number** of points by length and **Number** of points by height. If you press the **Apply** button, the procedure of surface formation by points specified on the screen will be started.

When you activate the **Extrude Surface** switch and click the **Apply** button, the running procedure displays a prompt on the command line

->Specify the base surface to be extruded или [?]:

It is necessary to specify the surface, then the window shown in Figure 24 will pop up on the screen.

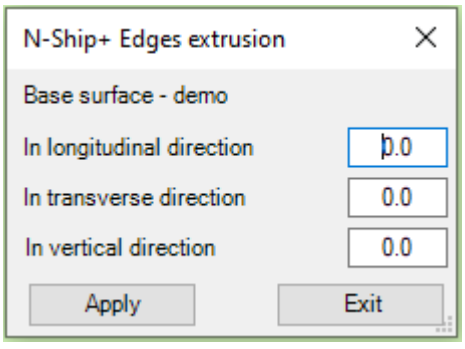


Figure 24. Edge extrusion dialog box

In this window it is necessary to specify in the input field the values of edge extrusion in the corresponding directions and click on the **Apply** button.

Attention: The directions are set relative to the WCS, longitudinal along the OZ axis, lateral along the OX axis and vertical along the OY axis. The value can be set with a + or - sign. All directions can be set at once. If for a given amount of displacement in the transverse direction towards the DP, the result is more than half the width of the ship's hull, the displacement will be up to the ship's DP.

The procedure outputs a request to the command line

->Specify the boundary edges or [?]:

You must specify the boundary edges of the base surface to be extruded.

When the **Default Surface** switch is activated, it is necessary to set the **Number** of points by length and **Number** of points by height in the input fields.

Attention: The number of points must not be less than the default number of points.

When you click the **Apply** button, the procedure of forming the frame of the SubDMesh similar to the one shown in Figure 25 is started.

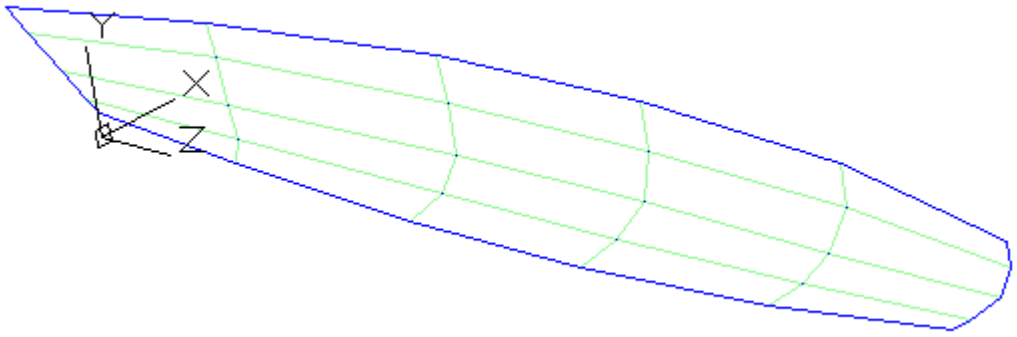


Figure 25. Formed surface frame

After obtaining the polyhedral network frame, you can start forming the ship surface of the required quality.

Attention: If the version of nanoCAD system you are using allows you to perform necessary manipulations with polyhedral network (smoothing, vertex editing, conversion to NURBS, etc.), you can use its capabilities, otherwise you need to use the toolkit, which is provided by the **Ship Surfaces** module....

Figure 26 shows the view of the function panel after the surface (polyhedral network) has been generated.

The tree shows the current surface and its objects – **Control grid** with sub-objects **Point**, **Rib** and **Edge**, as well as **Smooth network**.

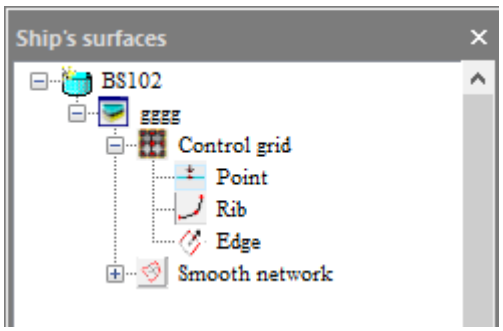


Figure 26. Function panel Ship's surfaces

When the **Control Grid** object is selected in the surface tree, the context menu contains the items – **Sections of grid**, **Show sections of grid** and **Hide sections of grid**, as shown in Figure 27.

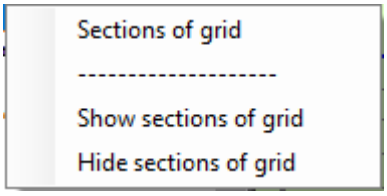


Figure 27. Function panel Control Grid

Selecting **Show sections of grid** or **Hide sections of grid** causes the corresponding actions with the control grid.

Selecting **Sections of grid** pops up the window shown in Figure 28.

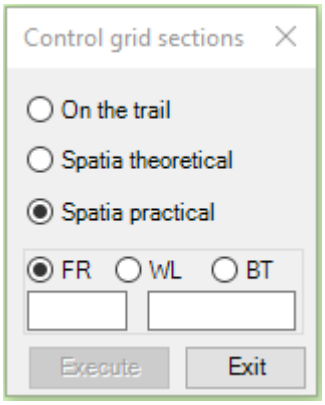


Figure 28. Network Control grid sections dialog box

In this window it is necessary to set the switch **Spatial theoretical** or **Spatial practical** and the switch **FR**, **WL** or **BT** and enter in the input fields the corresponding number and offset with a sign.

When activating the **By the trail** switch, the procedure of setting the position of the sowing on the screen is started.

Attention: You can use a point in the model to get the number and offset of the plane. The procedure of selecting a point is started by double clicking in the section number input field.

The context menu of the Point object is shown in Figure 29.

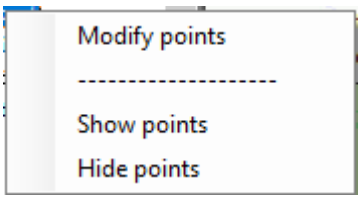


Figure 29. Context menu of the Point object

Selecting **Show points** and **Hide points** provides visualisation of the points.

Selecting the **Modify points** menu item brings up the window shown in Figure 30.

When you select the Edit menu item, the window shown in Figure 24 will pop up.

In this window the **Select**, **Split** and **Exit** buttons are available.

If you click on the **Split** button, the procedure of inserting a new point will be started, either in the specified point of the edge or in the middle of the edge, and the **Collapse** button will be unlocked, if you click on it, the procedure of removing the point inserted by the **Split** procedure will be started.

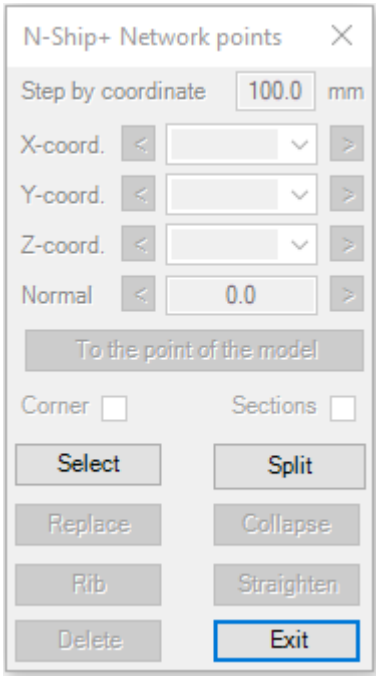


Figure 30. Network points

If you click on the **Select** button, you will start the procedure of point selection on the control grid.

Attention: You can select several points on the control grid at once. End the selection by **Enter** and **Esc**.

If a single point is selected, the window has the appearance shown in Figure 31, and on the reference grid the selected point changes colour to the colour of the selection from the **Installations** window.

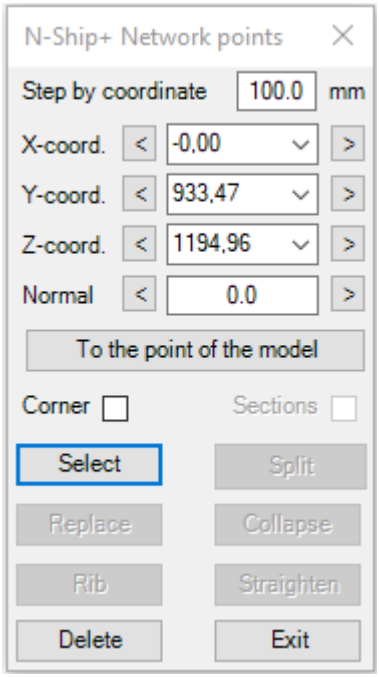


Figure 31. Network Points window when selecting a single point

In the upper part of the window there are unlocked the fields for input of parameters for changing coordinates of points, the Section pointer and the Delete button.

Attention: The Section pointer is only unlocked for regular points.

In the input field **Step by coordinate** you can specify the step of increment (positive number) of the corresponding coordinate or the value of displacement along the normal to the surface.

If you click on the buttons with '<' or '>' sign, the procedure of changing the corresponding coordinate or normal value by the specified step will be started, and the new point position will be displayed in the model as a point and a red line.

If you click on the **Normal** button, you will start the procedure of calculating the point that is distant from the selected point along the normal to the surface by the step value.

If you click on **To the point of the model** button, you will start the procedure of specifying the point in the model and all selected points will be assigned the coordinates of the point.

If you activate the **Corner** pointer, all selected points will be angular.

Any change of coordinates or pointers in the upper part of the window will unlock the Replace button.

If you activate the **Sections** pointer, the window shown in Figure 32 will pop up.

In this window, activate the pointers of the auxiliary cross sections to be generated on the smoothed network and click on the 'X' button.

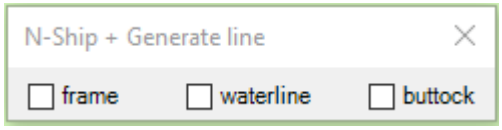


Figure 32. Generate line dialog box

Clicking on the Replace button starts the procedure of replacing the original point and when the Section pointer is active it will start the procedure of forming auxiliary sections (Frame, Waterline, Buttock) on the smoothed surface near the specified point, this helps to compare the result with the “marking lines” that can be inserted into the model.

Clicking on the Delete button starts the procedure of deleting the specified points.

Attention: When you delete a point, all faces that contain this point will be deleted.

When two points are selected, the Network Points window looks as shown in Figure 33.

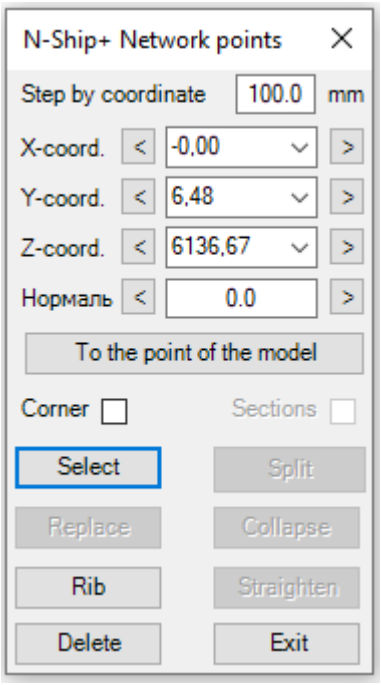


Figure 33. Network Points window when selecting two points

If you select two points, the **Rib** button will be unlocked and when you press it, the procedure of forming an edge between the specified points will be started.

When more than two points are selected, the Network Points window looks as shown in Figure 34.

If you click on the **Straighten** button, all the selected points will be moved to the line between the end points along the normal to the line.

Attention: If you select several points, the X, Y, Z coordinates of the points are entered in the drop-down lists X-coordinate, Y-coordinate, Z-coordinate. The number of lines in these lists is equal to the number of selected points and selecting any line will select the corresponding point and change the values in the lists to the coordinates of that point. For each point you can change the coordinates independently from the other points, and only when you use **Normal** and **To the point of the model** methods the changes will apply to all points.

Attention: You can return to the initial state using nanoCAD system rollback.

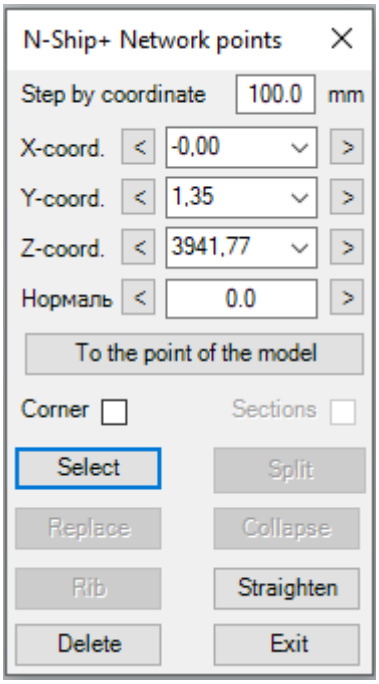


Figure 34. Dialog box when more than two points are selected

When you click on the **Exit** button you exit from the “N-Ship+ Network points.

When selecting the **Rib** object in the surface tree (Figure 26), the context menu contains the following items: **Breaking ribs**, **Delete ribs**, **Collapse ribs** and **Extrude the rib**.

Selecting the **Breaking ribs** item starts the procedure of forming a break line on the reference set - it is necessary to specify the edges that form the surface break. The color of these edges will change to the color of the **Ribs break Lines (control)** specified in the **Installations** window.

Attention: If you specify an existing break edge, it becomes a regular edge.

Selecting the **Delete ribs** item starts the procedure of deleting edges. This procedure can also delete edges of boundary lines.

Selecting the **Collapse ribs** item starts the procedure for deleting the specified regular edges or break lines from the control network.

Attention: The **Collapse ribs** item cannot be used to delete edges of boundary lines.

When selecting the **Extrude the rib** item, the window shown in Figure 35 pops up.

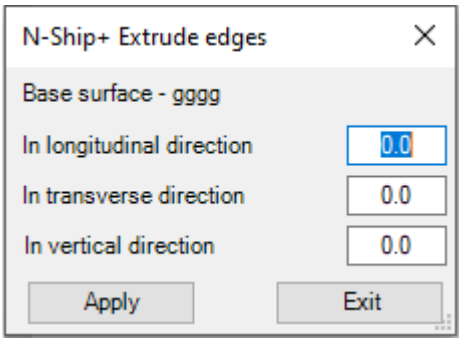


Figure 35. Extrude Rib dialog box

In this window you must set the amount of extrusion of the boundary line edge in the corresponding direction.

Attention: If for a given amount of displacement in the transverse direction towards the DP, the result is more than half the width of the ship's hull, the displacement will be up to the ship's DP.

Clicking the **Apply** button starts the procedure of rebuilding the control network of the surface with extruded edges of the boundary line.

When selecting the **Edge** object in the surface tree, the context menu contains the **Surface normals**, **Change normals** and **Hide normals** items.

When the **Surface normals** menu item is selected, the result is presented similarly to the one shown in Figure 36.

The reference network shows the normals in each face, all normals must point outward in the direction of the water.

When the menu item **Change normals** is selected, the direction of the normals is reversed as a result of the rearrangement of the reference network.

Selecting the **Hide normals** menu item delete the normals.

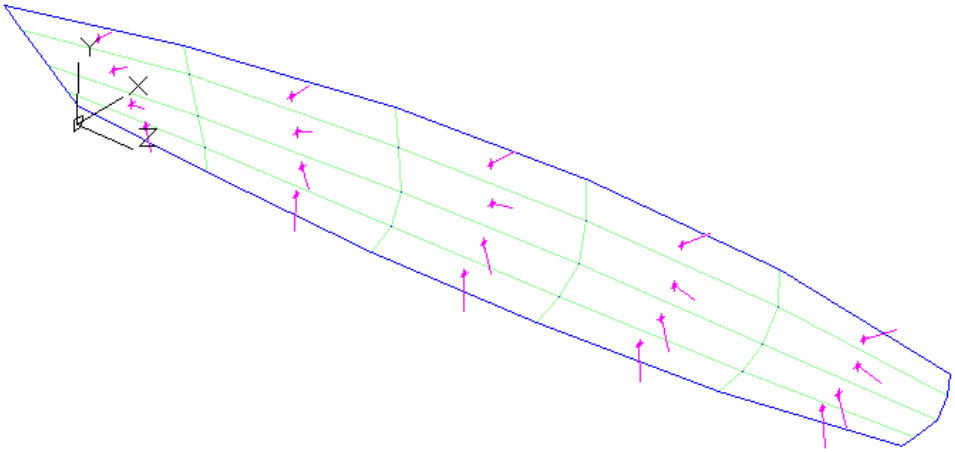


Figure 36. Reference network with normals

When selecting the **Smooth network** object in the surface tree, the context menu contains the following items: **Create surface**, **Delete surface**, **Show surface** and **Hide surface**.

When selecting the **Create surface** menu item, the procedure of control network smoothing is started.

The result is similar to the one shown in Figure 37.

The smoothed network is used to generate spar, waterline and bathos lines.

The menu items **Delete surface**, **Show surface** and **Hide surface** start the corresponding procedures.

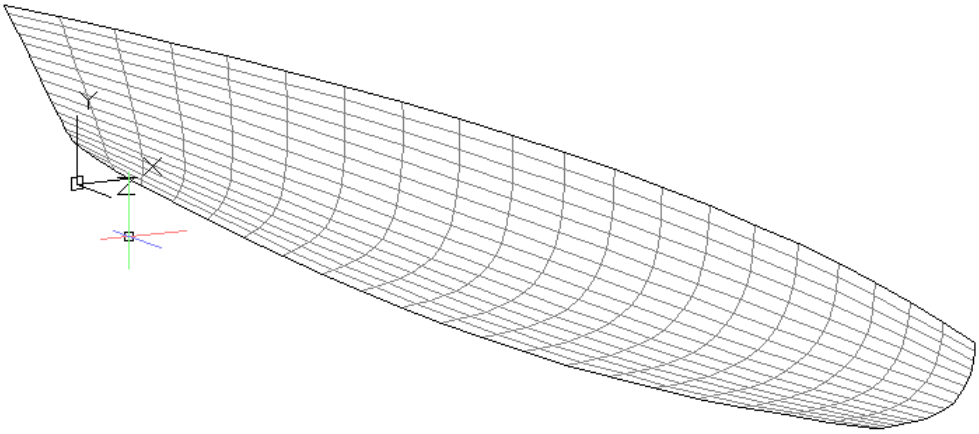


Figure 37. Smoothed network

When you select the **Create surface** menu item, the window shown in Figure 38 pops up.

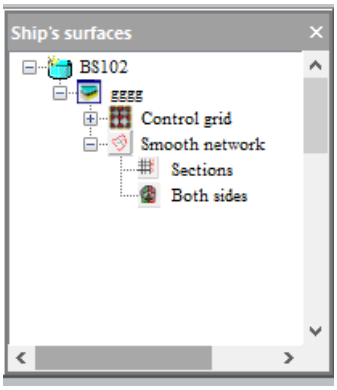


Figure 38. Function panel Ship's surfaces

Smoothed network is used to generate spars, waterlines and buttocks.

The context menu of the **Sections** object contains the items **Create sections**, **Delete sections**, **Show sections** and **Hide sections**.

When you select the **Create sections** menu item, the window shown in Figure 39 pops up.

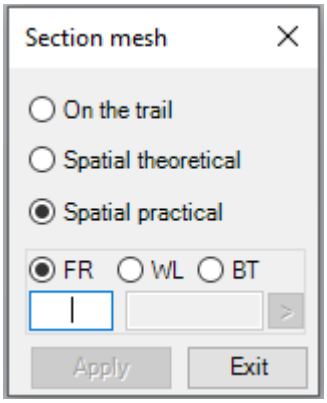


Figure 39. Dialogue window Section mesh

In this window it is necessary to set the switch **Spatial theoretical** or **Spatial practical** and the switch **FR**, **WL** or **BT** and enter in the input fields the corresponding number and offset with a sign, after which the button with the arrow “>” and the button **Apply** are unlocked.

If you double-click in the number input field, the procedure of specifying a point in the model is started, by which the number and offset of the corresponding section are determined.

If you activate the **On the trail** switch, the procedure of specifying the position of the sections on the screen is started.

If you click on the **Apply** button at once, one defined section will be formed.

If you click on the button with the arrow “>”, you can set the number and offset of the second section for the group of sections, and the button with the arrow “>” is replaced by the button with the arrow “<”, which will return to the previous window.

After setting the second section in the group (the process is similar to the one described above), the “<” arrow button will change to the “>” arrow button.

When you click on the “>” arrow button, the window shown in Figure 40 will pop up.

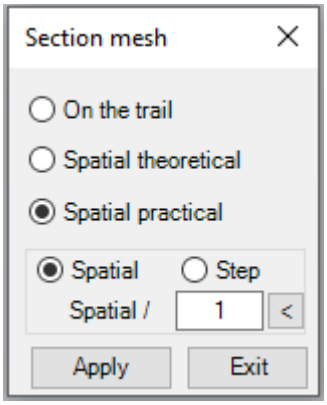


Figure 40. Dialogue window Section mesh

In this window it is necessary to set the parameter to form a group of sections of the control grid and click on the **Apply** button. The group of sections on the smoothed grid will be formed and return to the window shown in Figure 39.

If you click on the button with the arrow “<”, you will return to the window shown in Figure 39, but without calculating the section group.

The menu items **Delete sections**, **Show sections** and **Hide sections** are used to call the corresponding functions.

When selecting the **Both sides** object in the surface tree, the context menu contains the following items: **Create a second board**, **Delete a second board**, **Show a second board** and **Hide a second board**.

When selecting the **Create a second board** menu item, the procedure of symmetrization of the smoothed network and the existing sections with respect to the DP is started.

The **Delete a second board**, **Show a second board** and **Hide a second board** menu items are used to call the corresponding functions with respect to the second board.

When selecting the Save as fef file menu item (Figure 19), the window shown in Figure 41 pops up.

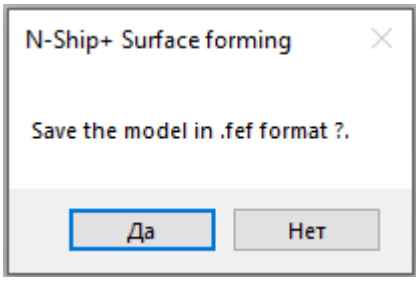


Figure 41. Save model dialog box

Clicking the **Yes** button starts the procedure for generating a file with the .fef extension and prompts you to place the file for saving.

5. WORK WITH MODELS

5.1. The window Interface of a command WORK ON MODEL…

The function panel called by the menu command **WORK ON MODEL …** is shown in Figure 42.

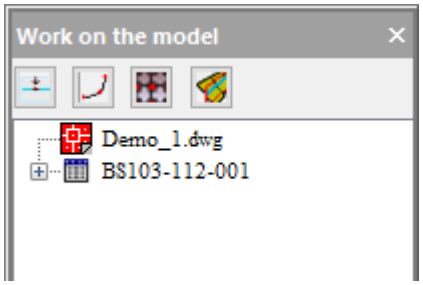


Figure 42. Function panel of a command **WORK ON MODEL …**

The panel contains a model tree in which the top node is the model file name, and below it are the drawings that are associated with that model.

There are five buttons at the top of the function panel:

 - button to call the points drawing window;

 - button to call the lines drawing window;

 - button to call the window of meshes building;

 - button to call the objects intersection window;

In the model objects like point, line, network or intersection lines of lines and surfaces with plane sections or between each other can be built in the windows called by clicking on the corresponding button in the upper part of the function panel.

5.2. Button Points

When the button  is clicked the “N-Ship+ Points” window shown in Figure 43 pops up.

The window has a **Selection points** toggle group with **All**, **Palette** and **Indication** toggles, a colour field and **Delete** and **Unite** buttons.

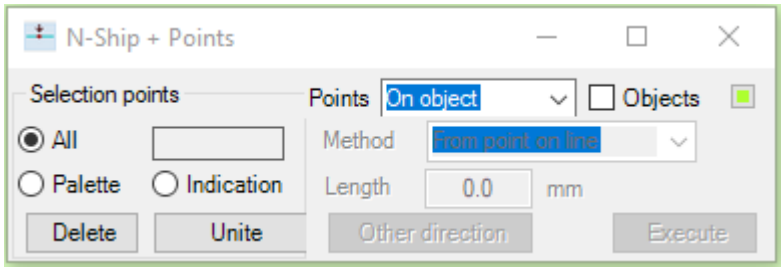


Figure 43. Dialogue window **Points**

The procedures for deleting and merging points, which are started by pressing the corresponding buttons, use the active switch in the **Selection points** group. Activate the **All** switch in the **Selection points** group to select all point type primitives in the model, the **Indication** switch to select primitives by the colour of the point specified in the model, and the **Palette** switch to select points by the colour selected from the palette. If you select points by colour, the selected colour is shown in the upper right corner of the **Selection points** group.

New points are generated by pressing the **Execute** and **Other direction** buttons. To activate these buttons, you must set the **Points** and **Method** of point generation. The **Points** drop-down list has the form shown in Figure 44.

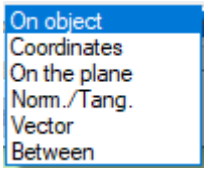


Figure 44. List of point formation types

The **Objects** pointer is unlocked when you select the method of setting the point **On object**.

Activating the **Objects** pointer activates the procedure for selecting objects in the model (line or network).

To the right of the **Objects** pointer there is a button  , when you click on which the selected model objects will be highlighted.

If lines have been selected, the **Method** list shown in Figure 45.

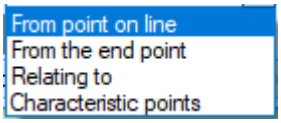


Figure 45. List of ways to generate points

If networks have been selected, the **Method** list shown in Figure 46 is activated.

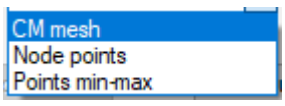


Figure 46. List of ways to form points on the mesh

If you select the **From point on line** method, in the **Length** field you can enter the distance in mm from the specified point to the calculated point. After specifying the length, the **Execute** and **Other direction** buttons are active. The procedure called by the **Execute** or **Other direction** button requests the point on the object and draws the calculated point on the current layer and in the current colour.

If you select the **From the end point** method, the procedure called by the **Execute** or **Other direction** button takes the start or end point of the line as the base point.

If you select the **Relating to** method, you must set the ratio of the length from the start point of the line to the calculated point to the length from the calculated point to the end point of the line. The relation 1:1 defines the midpoint of the line.

If you select the **Characteristic points** method, the **Create** list is activated, which has the form shown in Figure 47.

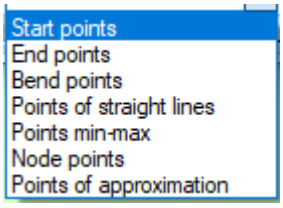


Figure 47. List of ways to form characteristic points

It is necessary to select in the list the type of points to be created and press the **Execute** button, the **Other direction** button is blocked in this case.

If you select the **CM mesh** method (centre of mass) click on the **Execute** button, the **Other direction** button is blocked in this case, the procedure of calculation and drawing the point of the centre of mass of the mesh starts.

If you select the method of setting the point **Norm./Tang.** (Figure 44) the “N-Ship+ Points” window will have the view shown in Figure 48.

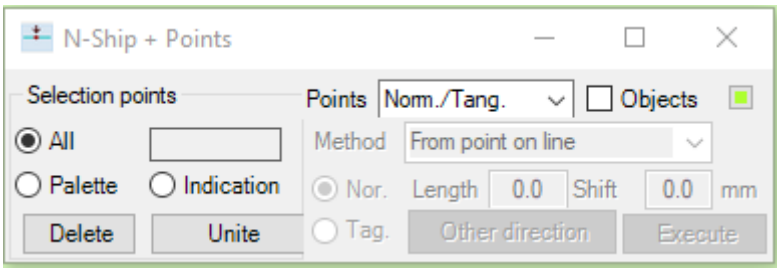


Figure 48. The dialogue box for forming points on **Norm./Tang.**

If you select **Norm./Tang.** point method, the **Objects** pointer will be unlocked.

If you select lines as **Objects** in the model, the **Method** list will be as shown in Figure 49.

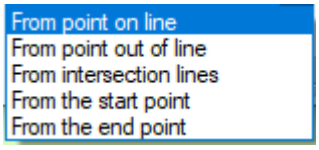


Figure 49. List of ways to plot points according to **Norm./Tang.**

If you activate the method **From point on line**, **From intersection lines**, **From the start point** or **From the end point**, you must enter the value of the normal / tangent in mm in the **Length** field. After that, the **Other direction** and **Execute** buttons are unlocked. The condition of solution search is set by the switches **Nor.** (normal) and **Tag.** (tangent) switches.

In the **Shift** input field you must set the distance along the line at which the point will be located, from which the point to be searched is defined by the normal. The length must be set with '-' sign, if the point must be shifted in the direction opposite to the line direction.

You can calculate the point by pressing the **Execute** button and by pressing the **Other direction** button to change the direction of the point calculation.

If you activate the method of defining **From point out of line**, the input field **Length** will be blocked because the origin point will be defined outside the object and the length of the normal/tangent will be determined by calculation.

The calculation of the point on the normal to the object is done by clicking on the **Execute** button.

If you select in the model of meshes, the list **Method** will have the form shown in Figure 50.

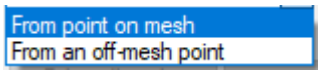


Figure 50. List of ways to calculate points on the mesh

If you activate the **From Point on mesh** method, the "N-Ship+ Points" window will look as shown in Figure 51.

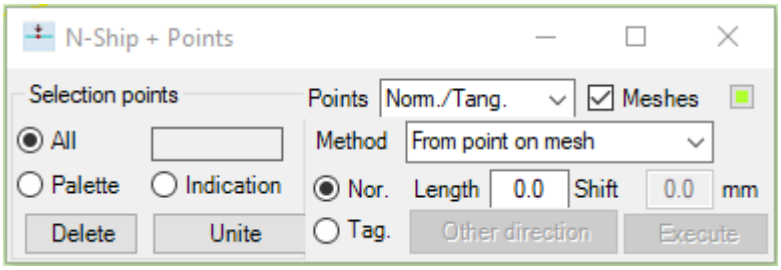


Figure 51. Dialogue window when selecting the **From Point on mesh**

It is necessary to enter the normal value in mm in the **Length** field, after that the **Other direction** and **Execute** buttons will be unlocked.

If you activate the method of defining **From an off-mesh point**, the input field **Length** will be blocked, because the reference point will be defined outside the network (the normal length will be determined by calculation). The procedure of point calculation is started by clicking on the **Execute** button.

If you select the method of point definition **Coordinates** (Figure 44), the window "N-Ship+ Points" will have the view shown in Figure 52.

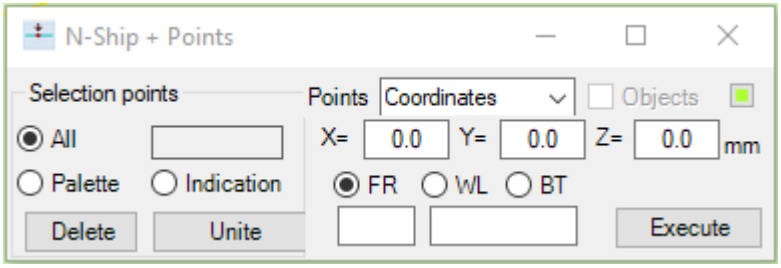


Figure 52. Dialogue window when specifying the coordinates of a point

In this window it is possible to specify in the corresponding input fields the coordinates of the point, at the same time you can use the table of spatia, specifying the number and offset for the corresponding section (Frames - FR, Waterlines - WL, Buttocks - BT). When you click on the **Execute** button, the procedure of calculation and drawing of the point with the given coordinates is started.

If you select the method of specifying a point **On the plane** (Figure 44), the “N-Ship+ Points” window will have the view shown in Figure 53.

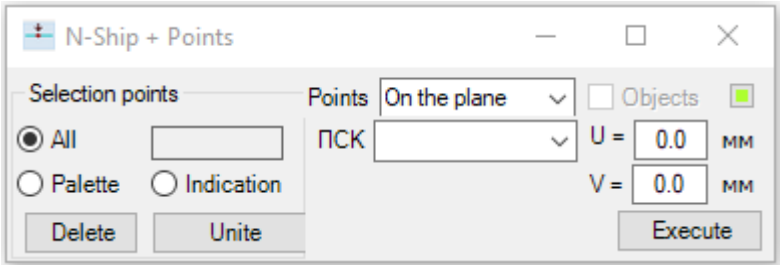


Figure 53. Dialog box when specifying a point in the plane

In this method the point is defined by two coordinates - U and V in the corresponding plane. The plane is defined by selecting from the UCS list shown in Figure 54.

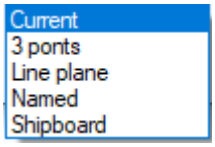


Figure 54. UCS list

Current - current user coordinate system, **3 points** - user can specify the coordinate system by specifying 3 points, **Line plane** - the plane of the specified plane line is used.

If you specify **Named** - the list of named user coordinate systems is expanded, from which you can make a selection.

If you specify **Shipboard** - you can specify a coordinate using the spacing table. In this case the corresponding coordinate will be converted to the set user coordinate system.

You can draw the point by clicking on the **Execute** button.

If you select the **Vector**. (Figure 44) the “N-Ship+ Points” window will look as shown in Figure 55.

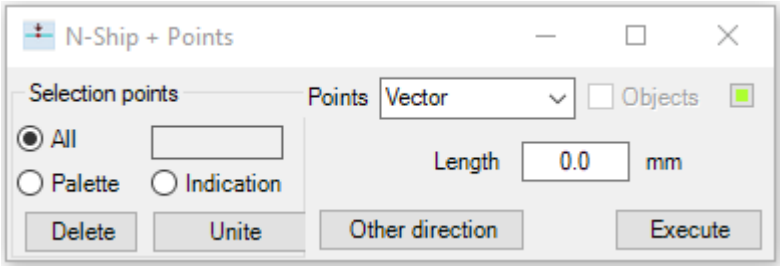


Figure 55. Dialog box when selecting a **Vector** point

In the input field **Length** sets the length of the vector in mm, i.e. at what distance in the given direction from the given point it is necessary to build the desired point. The **Other direction** button changes the direction to the opposite one.

If you select the **Between** point setting method (Figure 44), the “N-Ship+ Points” window will have the form shown in Figure 56.

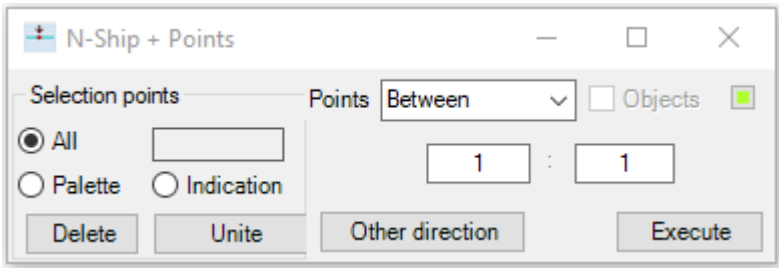


Figure 56. Dialog box when selecting a point **Between**

In the input fields you specify the ratio of the segments along the length of the segment from the first specified point to the second specified point. If you click the **Other direction** button, the direction changes to the opposite direction.

5.3. Button Lines

When you press the button  pops up a window that looks like the one shown in Figure 57.

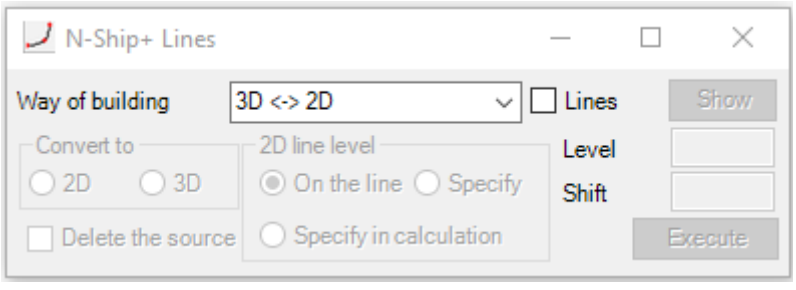


Figure 57. **Lines** dialog box

The **Way of building** drop-down list provides the ways to define the lines shown in Figure 58.

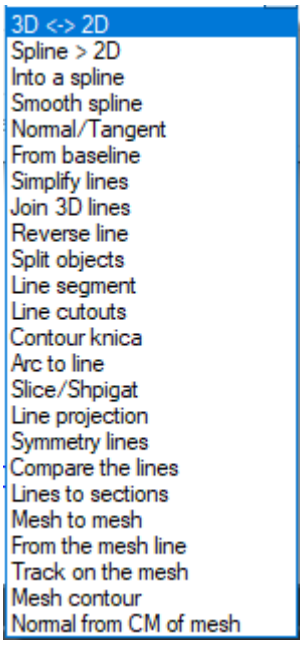


Figure 58. List of ways to define lines

The **3D <-> 2D** plotting method is intended for converting lines into either flat (2D) or spatial (3D) lines, which is set by the corresponding switches in the **Convert to** group.

Activation of the **Lines** pointer starts the procedure of line selection in the model. Pressing the **Show** button starts the procedure of highlighting the selected lines in the model.

For lines of 2D type, the 2D line **Level** selection field and the field for entering the value of shift with sign (plus or minus) of the resulting line relative to the plane of the original line are unlocked. If you activate the **On the line** switch (active by default), the procedure launched by the **Execute** button determines the position of the source line plane and displays a message if the line does not lie in the plane. If you activate the **Specify** switch, the procedure requesting the point lying on the object is started and the current level is set by the Z coordinate of the specified point in the current coordinate system, which is shown in the Level input field. If you activate the switch **Specify in calculation**, then the procedure started by the **Execute** button, during the calculation process, requests a point on the model object by the Z coordinate of which the current level is set in the current coordinate system and the line is generated.

In the Shift input field you can set the value of level shift with sign relative to the current level.

Activating the **Delete the source** pointer indicates that the source lines are to be deleted.

The line calculation procedure is started by pressing the **Execute** key.

The **Spline > 2D** construction method is designed to convert splines into 2D lines and the “N-Ship+ Lines” window has the form shown in Figure 59.

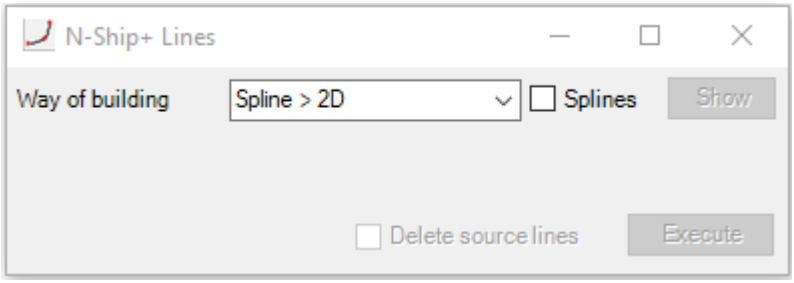


Figure 59. Dialog box for construction method **Spline > 2D**

Only splines can be used as objects. If the initial spline is flat, the resulting 2D-polyline will lie in the plane of the spline, if the spline is spatial, the screen displays the query shown in Figure 60.

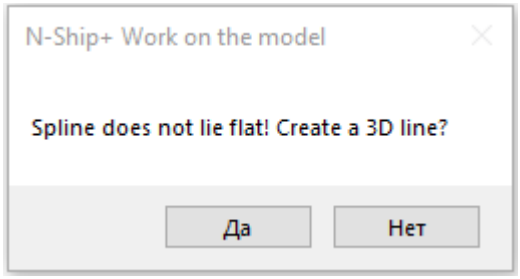


Figure 60. Request for 3D line creation

Pressing the **Yes** key generates a 3D polyline.

The method of construction **Into a spline** is designed to convert lines into spline and the window “N-Ship+ Lines” has the form shown in Figure 61.

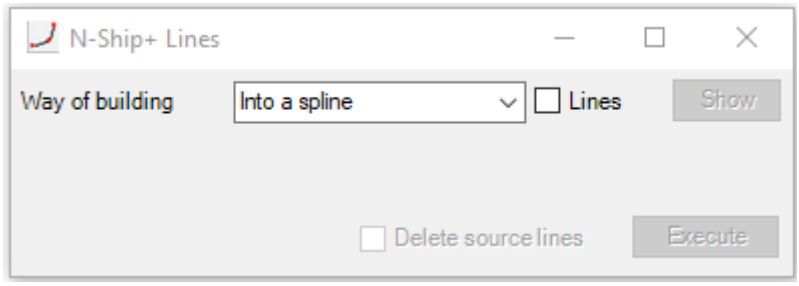


Figure 61. Dialog box for building method **Into a spline**

Construction method **Smooth spline** is intended for calling the function of smoothing the spline and drawing the curvature graph, the “N-Ship+ Lines” window has the form shown in Figure 62. Only splines can be used as objects. After selecting objects of spline type it is necessary to set the value of maximum departure of reference points during smoothing in the **Max** input field and start the smoothing procedure by the **Smooth** button.

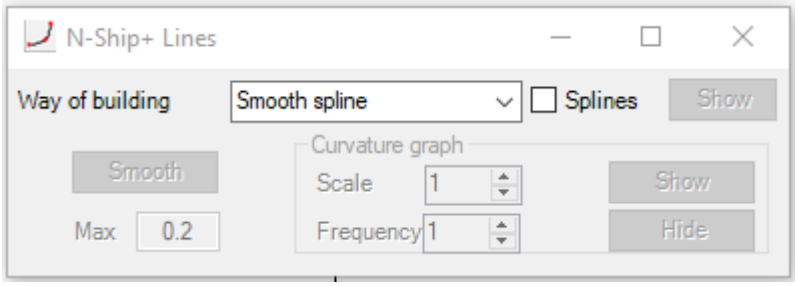


Figure 62. The dialog box for the **Smooth spline** method

The **Show** and **Hide** buttons are used to plot the curvature graph.

Using the “up-down” controls **Scale** and **Frequency** set the parameters of the curvature graph.

When **Normal/Tangent** plotting method is selected, the “N-Ship+ Lines” window looks as shown in Figure 63.

The construction method is used to generate a normal/tangential segment to the specified objects depending on the activation of the **Norm.** or **Tang.** pointers. The objects selected in the model when the **Lines** pointer is activated can be either lines or polygonal meshes.

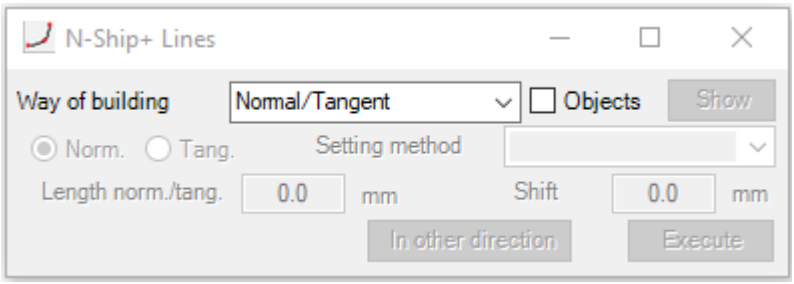


Figure 63. **Normal/Tangent** method dialog box

After selecting objects of the line type, it is necessary to **Setting method** of assignment from the list shown in Figure 64.

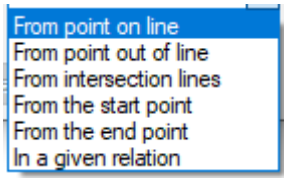


Figure 64. List of ways to specify line drawing

If you select objects of the Mesh type, **the Setup method** list will look as shown in Figure 65.

Figure 65. List of ways to specify the line construction when selecting a mesh

If you activate the method of defining **From point on mesh**, you must enter the normal value in mm in the **Length** field, after that the **Other direction** and **Execute** buttons will be unlocked. By default, the point is drawn along the normal to the object.

The calculation of the point is done by clicking on the **Execute** and **Other direction** buttons.

If you activate the **From an off-mesh point** method, the input field **Length** will be locked, because the origin point will be defined outside the mesh, the normal/tangent length will be determined by calculation. The calculation of the point is done by clicking on the **Execute** button.

If you activate the methods of defining **From point on line**, **From intersection lines**, **From start point** or **From end point**, the window looks as shown in Figure 66

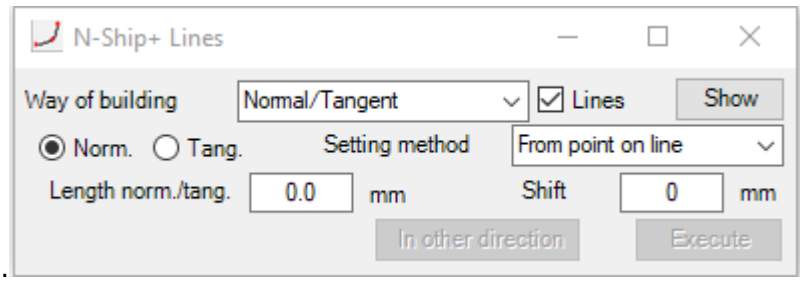


Figure 66. Dialog box for the method of specifying from a point on a line

In the **Length norm./tang.** input field you specify the length of the normal or tangent respectively. In the **Shift** input field, you can specify the distance along the line from the point with a sign that is indicated on the line. The point is always searched along the line direction, if the value of the parameter is set with minus, the point is taken against the line direction.

The calculation is performed by clicking on the **Execute** button. If you want to change the direction of the calculated segment, you must click on the button **Other direction**.

If you activate the **From point out of line** method, the window looks as shown in Figure 67.

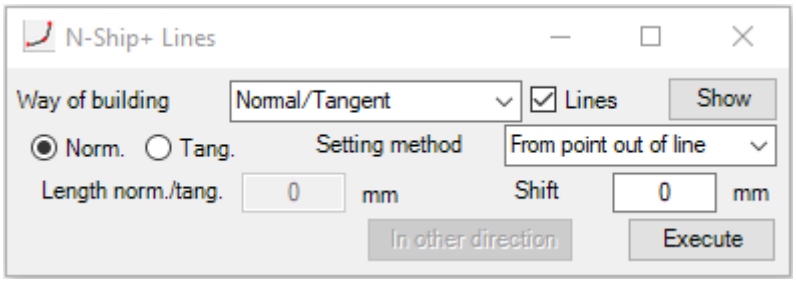


Figure 67. Dialog box for **From point out of line** method

The input field **Length norm./tang.** is locked. Calculation is performed by pressing the **Execute** key or the **Other direction**. If the procedure cannot find a solution, a warning is displayed.

When you activate the **In a given relation** to sections method, the window looks as shown in Figure 68.

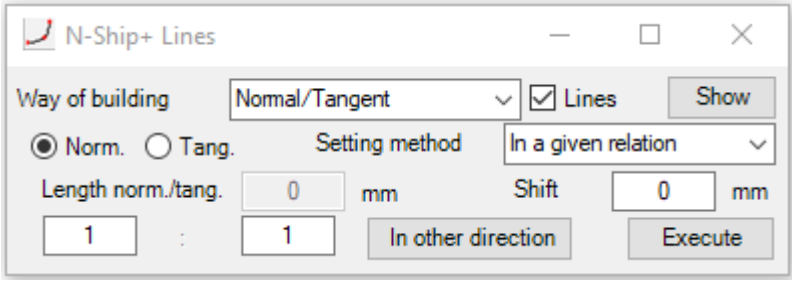


Figure 68. Dialog box for assignment method **In a given relation**

The starting point of the tangent or normal will be obtained at a distance from the starting point of the line obtained as the result of dividing the length of the line from the starting point to the end point of the line by a number equal to the sum of the numbers set in the relation input fields multiplied by the number set in the first input field.

For example, if 1 is set in the first input field and 1 is set in the second input field, the line is divided in half. If you enter 2 in the first input field and 1 in the second input field, the line is divided into three sections and the normal origin point is located at the length of 2 sections from the initial point of the line.

If you click on the **Other direction** button, the normal origin point will be on the length of the number of segments defined in the second input field.

For all methods of point setting, the calculation is performed by clicking on the **Execute** button. If you click on the **In other direction** button, the direction of the resulting line will be reversed.

If you select the **From baseline** method, the window looks as shown in Figure 69.

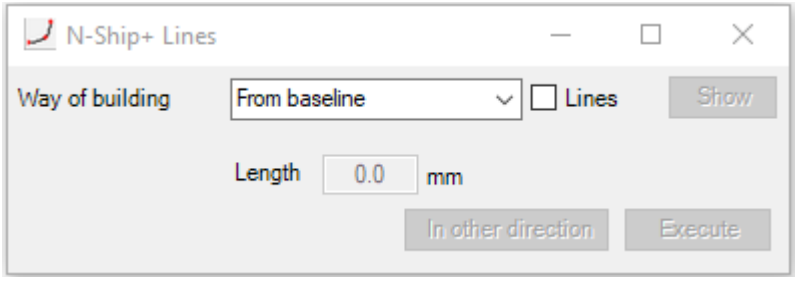


Figure 69. Dialog box for **From baseline**

Drawing method **From baseline** is intended for drawing a line on a set of lines, selection of which is made in the model by activating the **Lines** pointer, after which the **Other direction** and **Execute** buttons will be unlocked. In the input field **Length** you must specify the distance from the intersection points of the set of lines with the line, the selection of which is made when you click on the **Execute** button. If you want to draw a line from the other side of the base line, you must click the **Other direction** button.

If you select the **Simplify lines** method, the window looks as shown in Figure 70.

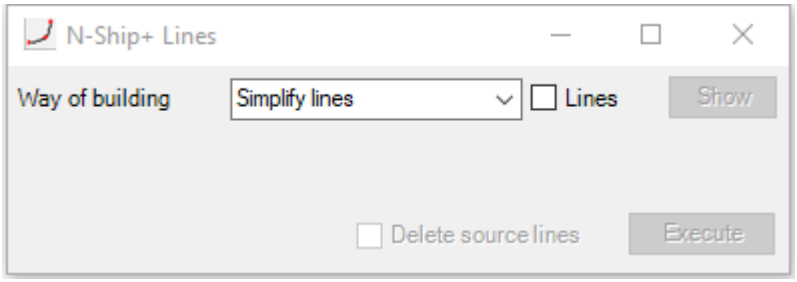


Figure 70. Dialog box for **Simplify lines**

When the **Lines** pointer is activated, the procedure of selecting lines in the model is started, after which the **Execute** and **Delete source lines** buttons will be unlocked.

When you click on the **Execute** button, the simplification procedure for lines will be started. Simplification consists in searching for points lying on rectilinear segments and excluding them from the resulting line.

Activation of the **Delete source lines** pointer leads to deleting the original lines after simplification.

If you select the **Join 3D lines** construction method, the window looks as shown in Figure 71.

When the Lines pointer is activated, the procedure of selecting lines in the model is started.

If you press the **Execute** key, the procedure of merging 3D lines is started.

Activating the **Delete source lines** pointer deletes the original lines after merging.

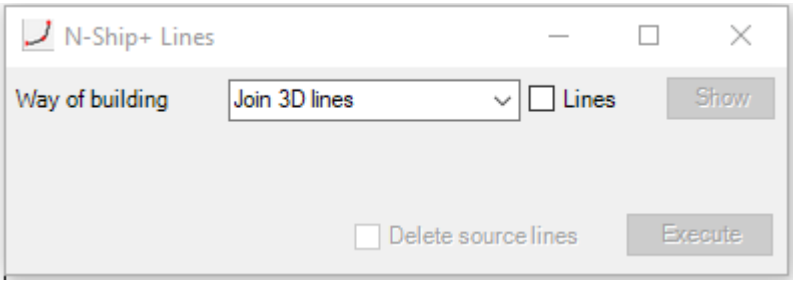


Figure 71. Dialog box for **Join 3D lines**

If you select the **Reverse line** method, the window has the view shown in Figure 72.

Figure 72. Dialog box for **Reverse line**

The **Reverse line** method is used to change the direction of lines that are selected from the model by activating the **Lines** pointer.

If you press the **Execute** key, the lines with the start and end points reversed are calculated.

Activating the **Delete source lines** pointer deletes the original lines.

If you select the **Split objects** construction method, the window looks as shown in Figure 73.

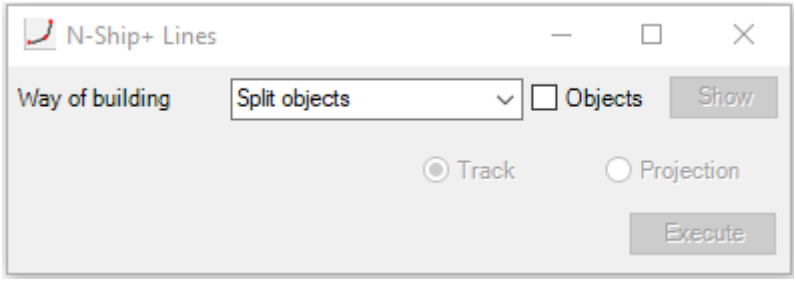


Figure 73. Dialog box for **Split objects**

The **Split objects** method is used to cut lines, polygonal meshes and bodies, which are selected from the model when the **Objects** pointer is active.

Attention: Only lines and bodies or only polygonal meshes can be specified as objects when the Track switch is active, and only lines can be specified as objects when the **Projection** switch is active.

When you press the **Execute** key when the Track switch is active, the procedure of cutting the selected objects by a plane, the trace of which is defined by two points, is started, and when the

Projection switch is active, the procedure is started, which in the command line gives a request to choose a line, the projection of which relative to the current screen will define the intersection surface of the set of objects. After the procedure is executed, you can delete those parts of the lines that are not needed, as well as delete the trace or projection line.

If you select the method **Line segment**, the window looks as shown in Figure 74.

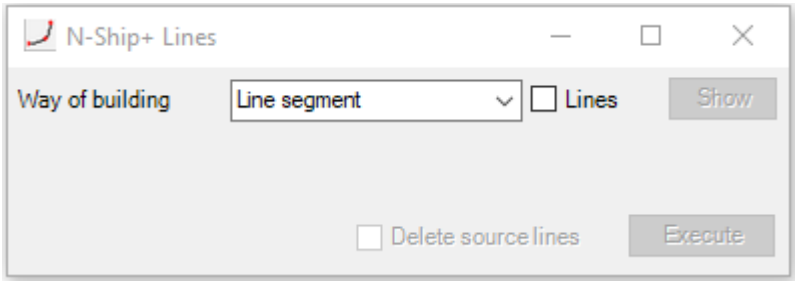


Figure 74. Dialog box for **Line segment**

Method of drawing **Line segment** is intended for selecting the line segment, which are selected from the model when activating the Line pointer. Boundary conditions are requested when executing the procedure from NanoCAD command line. If you specify - **Points**, the boundaries of the line segment are defined by the specified points, if you specify - **Lines**, they are defined by the specified lines.

The **Delete source lines** pointer is intended for deleting, if necessary, source lines from the model. The calculation is performed by clicking on the **Execute** button.

If you select the method **Line cutouts**, the window looks as shown in Figure 75.

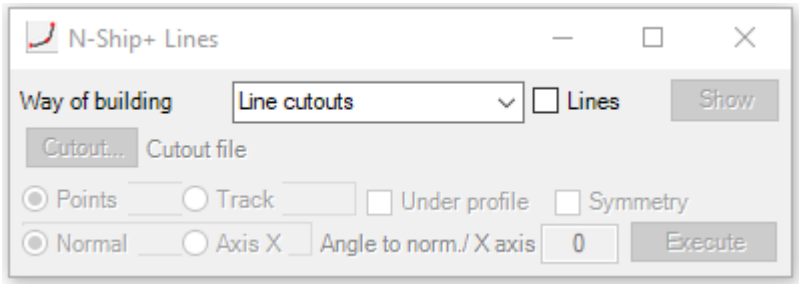


Figure 75. Dialog box for **Line cutouts**

The **Line cutouts** method is used to create a cutout on lines that are selected from the model by activating the **Lines** pointer. When you click on the **Cutout...** button, the window shown in Figure 76.

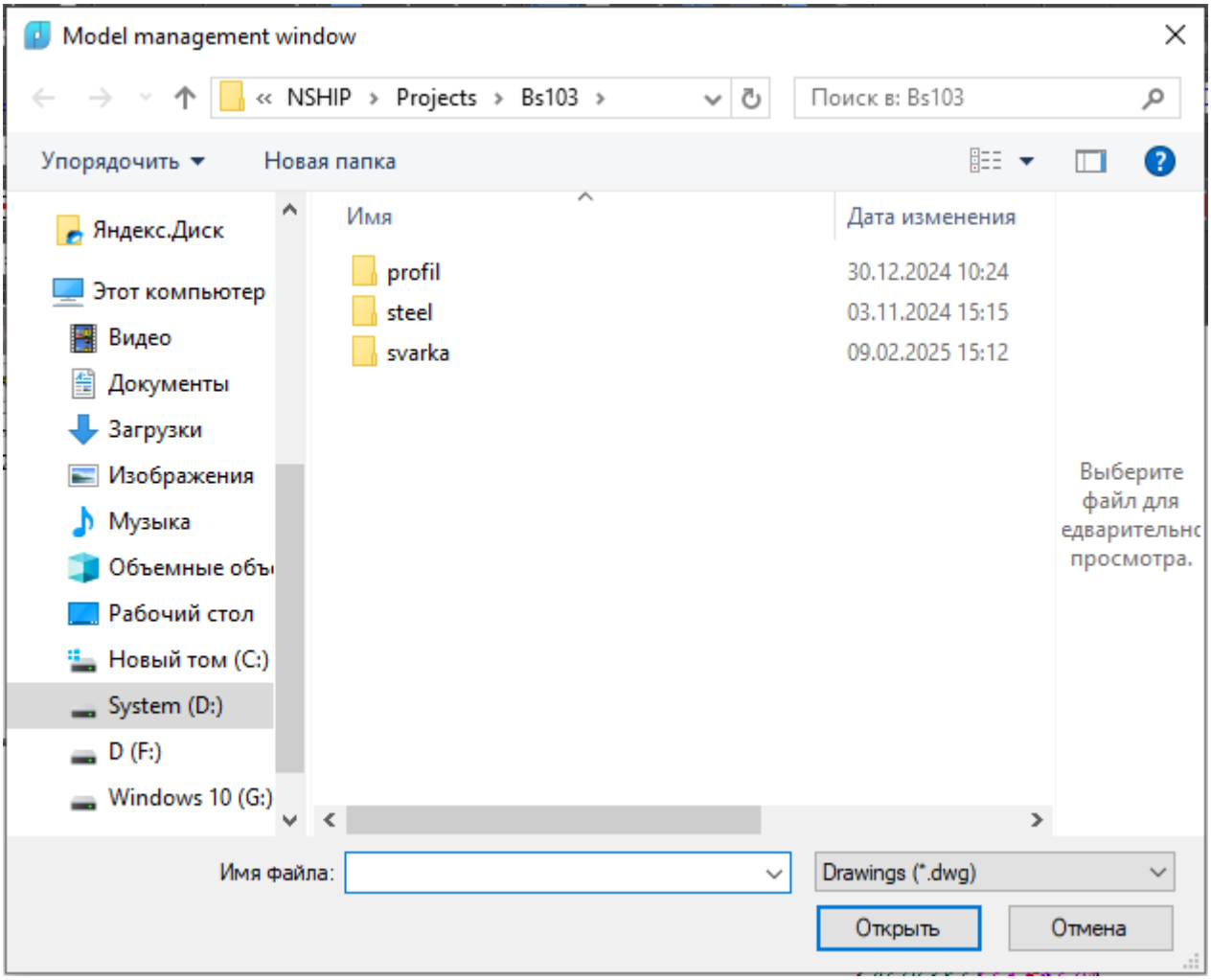


Figure 76. Select dwg file dialog box

In this window it is necessary to set the access path, select the cutout file and click the **Open** button. After that the window will look as shown in Figure 77.

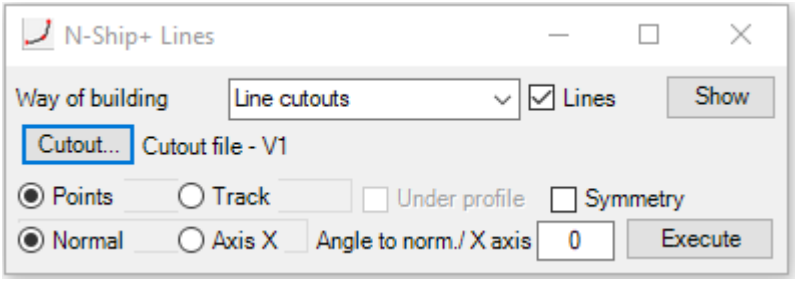


Figure 77. Dialog box after selecting the dwg file of the cutout

Activating the **Points** switch means that the cutout is built on the lines at the specified points.

Activation of the **Track** switch means that the cutout is built at the point of intersection of lines with the specified trace. Activating the **Track** pointer will unlock the **Under profile** pointer.

Activation of the **Under profile** pointer means that when performing the procedure of cutout construction at the intersection points with the trace, the transformation of cutouts under the profile passage

along the trace line will be carried out, at that the angle of cutout construction is determined by the trace plane and the switches **Normal** and **Axis-X**, as well as the field of angle input will be locked.

Activation of the **Normal** switch defines the condition of binding the cutout by normal to the line, and activation of the **Axis-X** switch to the axis X of the world coordinate system. In the **Angle to normal/X-axis** input field you can set the angle of deviation from the reference axis with a sign, if necessary.

If the **Symmetry** pointer is activated, the cutout will be symmetrized relative to the reference axis.

Calculation is performed by clicking on the **Execute** button.

If you select the method of building the **Contour knica**, the window looks as shown in Figure 78.

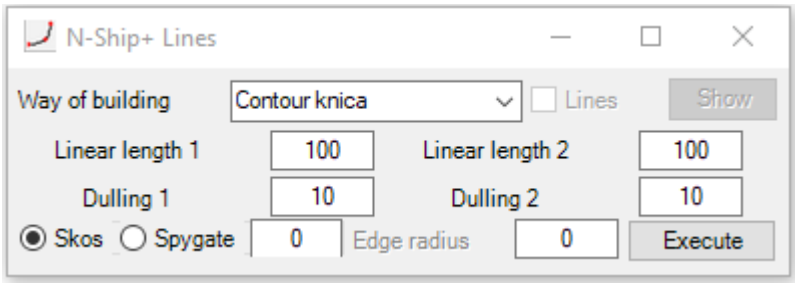


Figure 78. Dialog box for **Контур кницы**

Attention: The **Lines** pointer is not used in this procedure. The procedure can only be applied to flat lines lying in the same plane.

In the corresponding input fields it is necessary to set the parameters of the line.

If you press the **Execute** key, the procedure of drawing the contour of the book on the lines requested by the drawing procedure is started.

If you select the **Arc to line** method, the window looks as shown in Figure 79.

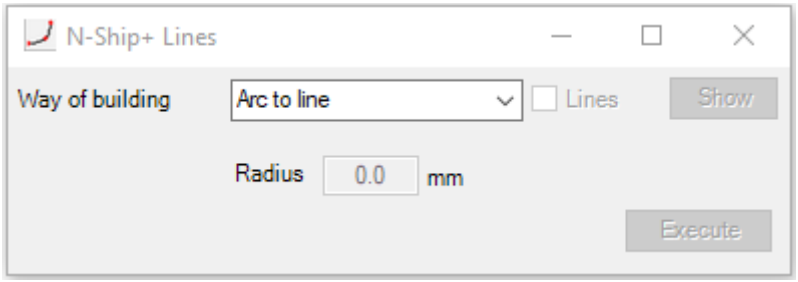


Figure 79. Dialog box for **Arc to line**

The procedure of drawing an arc of a given radius from a specified point can be applied only to flat lines. When you press the Execute key, the procedure asks for a point and a line. If it is possible, the procedure draws two arcs, the unnecessary one must be deleted.

If you select the **Slice/Spigat** method, the window looks as shown in Figure 80.

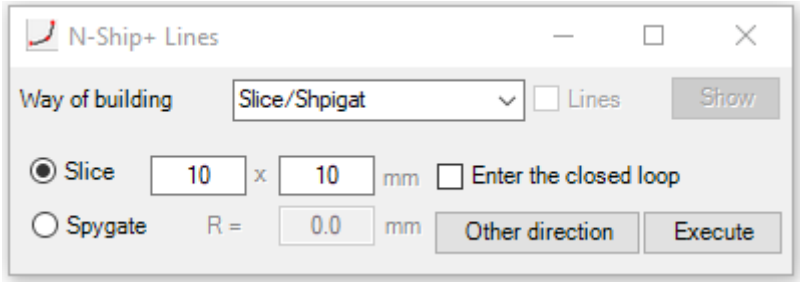


Figure 80. Dialog box for **Slice/Spigat**

The **Slice/Spigat** method is used to draw a cut or spigat line between the lines that are selected in the model. Depending on the active **Slice** or **Spigate** switch, the corresponding fields for entering parameters are activated.

You can set different values for the slice in the input fields and if the result of the **Execute** button is not satisfactory, you can recalculate the slice by pressing the button **Other direction**.

For the scupper the calculation is performed by clicking on the **Execute** button.

If you select the method of construction **Lines projection**, the window looks as shown in Figure 81.

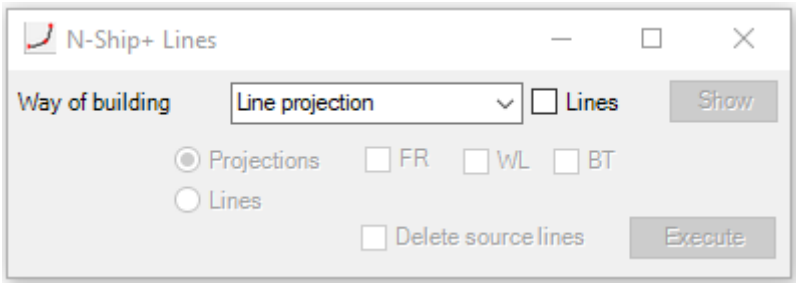


Figure 81. Dialog box for **Lines projection**

When the **Lines** pointer is activated, the procedure of selecting lines in the model is started.

The **Delete source lines** pointer is used to specify the deletion of source lines.

The selection of the construction procedure is set by the **Projections** and **Lines** switches. It is possible to generate either spatial lines, for which two projections are specified, or projections, if spatial lines are specified. The type of projections is defined by the corresponding pointers FR (frame), WL (waterline), BT (buttock).

If you press the **Execute** button, the line drawing procedure is started.

If the **Symmetry lines** construction method is selected, the window looks as shown in Figure 82.

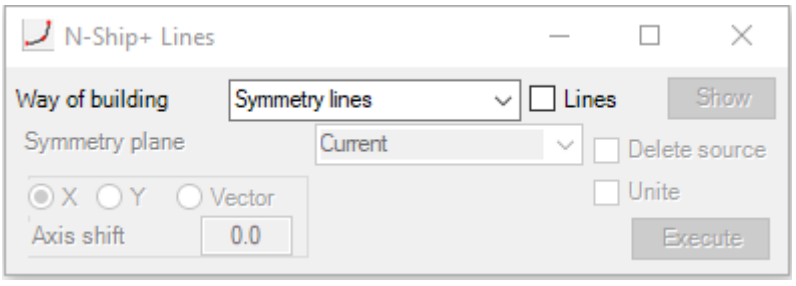


Figure 82. Dialog box for **Symmetry lines**

Symmetry lines method is intended for construction of symmetry lines for lines selected in the model when activating the **Lines** pointer.

Symmetry plane construction is selected from the list shown in Figure 83.

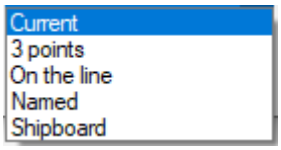


Figure 83. List for symmetry plane selection

Current - plane of the current UCS.

3 points - to define the plane by 3 points in the model.

On the line - to define the plane by the flat line specified in the model.

Named - to select the plane of the named UCS. The list of named UCS is opened when you select this method.

Shipboard - to define the symmetry plane using information from the spars table by the number and offset of the spar, waterline or buttock, respectively.

The **X** and **Y** switches are used to set the corresponding symmetry axes in the selected symmetry plane. The symmetry axis can be shifted by the distance specified in the input field with + or - sign.

The **Vector** switch allows to set the symmetry axis by specifying 2 points on the symmetry plane in the model during calculation.

When the **Delete source** pointer is activated, the source lines are deleted.

If the **Unite** pointer is activated, the source and symmetrical lines are merged into one.

The symmetrization procedure is started by pressing the **Execute** button.

If you select the **Compare the lines** method, the window looks as shown in Figure 84.

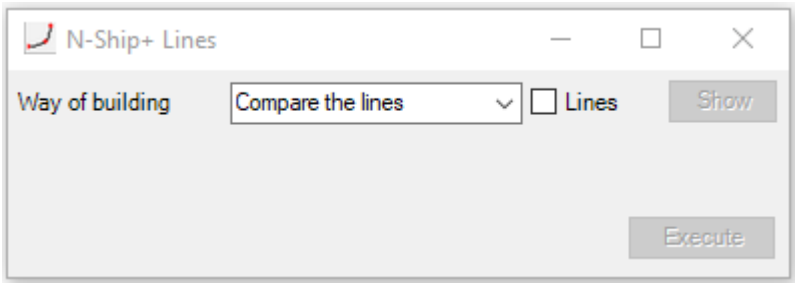


Figure 84. Dialog box for **Compare the lines**

When the **Lines** pointer is activated, the procedure of selecting lines in the model is started. It is necessary to specify two lines for comparison.

On the button **Execute** the calculation is performed and on the line is drawn a point maximally distant from a similar point on the other line, and the window displays “Maximum distance at the specified point = (value)!” in mm.

If you select the method of drawing the **Line on sections**, the window looks as shown in

Figure 85.

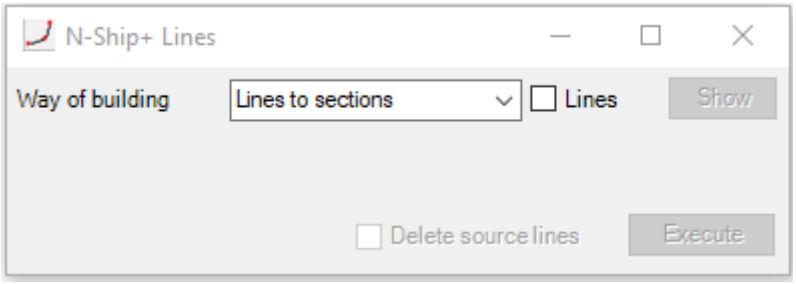


Figure 85. Dialog box for **Line on sections**

When the **Lines** pointer is activated, the procedure of selecting lines in the model is started. It is necessary to specify the lines to be transferred to a set of flat lines (e.g. frames).

The **Delete source lines** pointer is used to specify the deletion of the original lines.

Clicking on the **Execute** button starts the calculation procedure, which asks on the command line *->Specify the sections to which the lines are to be transferred:*

If you select the **Mesh to mesh** construction method, the window is shown in Figure 86.

The **Mesh to mesh** method is intended for transferring a line from one mesh to another. Meshes must have the same dimensions M and N. The lines that are selected by activating the **Lines** pointer must have points that lie on the edges of the mesh.

When the **Execute** key is pressed, the calculation procedure asks for the source mesh and the result mesh.

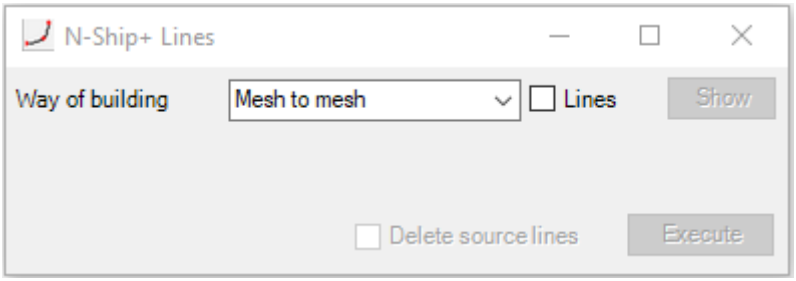


Figure 86. Dialog box for **Mesh to mesh**

When **From the mesh line** is selected, the view of the window is shown in Figure 87.

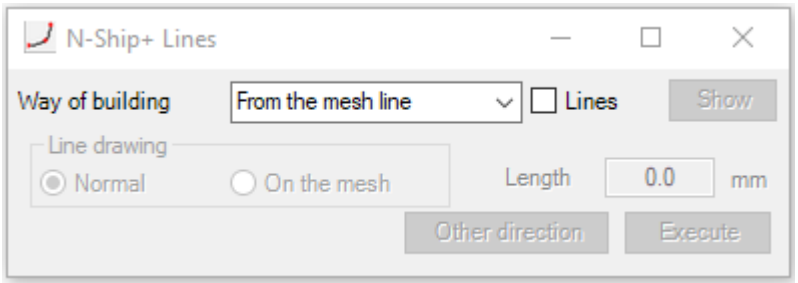


Figure 87. Dialog box for **From the mesh line**

The **From the mesh line** method is used to calculate the line at the distance specified in the **Length** input field from the specified line along the normal to the network, if the **Normal** pointer is activated, or along the N direction of the mesh (MxN), if the **On the mesh** pointer is activated.

When the **Lines** pointer is activated, the procedure of selecting the lines lying on the network is started.

If you press the **Execute** button, the procedure of line calculation is started. If you want to draw the line from the other side, you must click on the **Other direction** button.

If you select the method of construction **Track on the mesh**, it has the form shown in Figure 88.

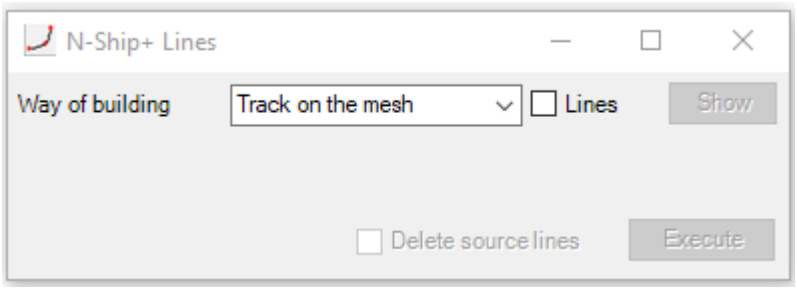


Figure 88. Dialog box for **Track on the mesh**

The method of constructing Track on the mesh is intended to calculate the points of intersection of the line with the mesh faces and to form a new line with inclusion of all intersection points in it. It is assumed that the initial line lies on the surface of the mesh.

When the **Lines** pointer is activated, the procedure of model lines selection is started.

The **Delete source lines** pointer should be activated if you want to delete source lines.

If you click on the **Execute** button, the procedure of line calculation in the model is started.

If you select the **Mesh contour** method, the window has the view shown in Figure 89.

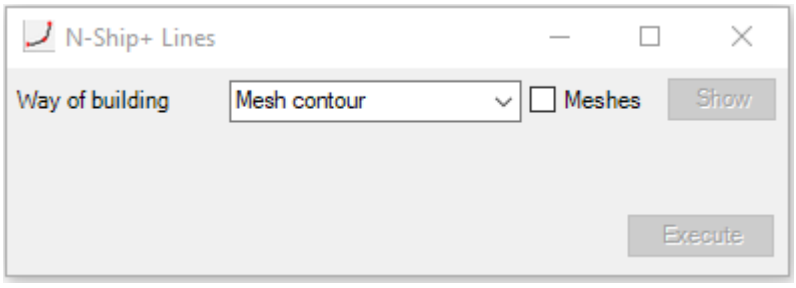


Figure 89. Dialog box for **Mesh contour**

Construction method **Mesh contour** is intended for construction of mesh contours in the form of spatial lines. The **Meshes** pointer starts the procedure of meshes selection in the model.

When you click the **Execute** button, the lines of mesh contours are drawn.

If you select the **Normal from CM of mesh**, the window looks as shown in Figure 90.

The **Normal from CM of mesh** method is intended for calculation and construction of the normal from the center of mass of the network with the length specified in the input field **Length**.

The **Meshes** pointer starts the procedure of selecting meshes in the model.

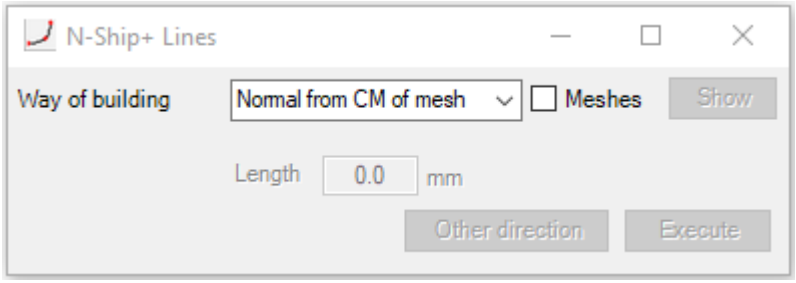


Figure 90. Dialog box for **Normal from CM of mesh**

If you click the **Execute** button, the normal is drawn, if you click the **Other direction** button, the normal is drawn in the other direction.

5.4. Button meshes

When you press the button  the window shown in Figure 91 pops up.

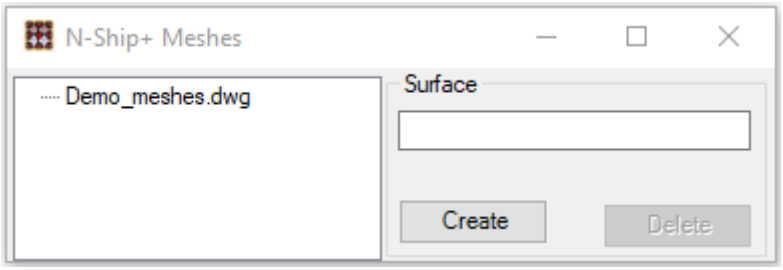


Figure 91. Dialog box for **Meshes**

The tab window contains the window of the surfaces tree and the Surface group with the field for entering the surface name and the **Create** and **Delete** buttons.

The **Create** and **Delete** buttons are used to create or delete, respectively, named items in the surface tree.

Attention: Surface is a named group of meshes, so deleting a mesh from a group does not lead to deleting the mesh itself, the deletion of the net must be done by nanoCAD tools.

In nanoCAD there are no tools for building meshes for N-Ship+ (polygonal meshes). But there is a set of tools for building 3D surfaces. It is necessary to build the necessary number of sections on the surface, and then build a mesh using the **Mesh from lines** function.

In the input field set the name of the surface and click the **Create** button. The procedure of model surface tree formation starts, the window is shown in Figure 92.

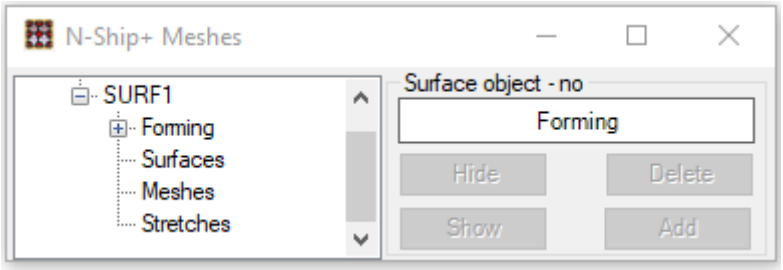


Figure 92. Surface tree dialog box

Four branches are automatically created in the tree under the item with the surface name:

Forming, **Surfaces**, **Meshes** and **Stretches**.

When you select **Forming** in the object tree, the window looks as shown in Figure 93.

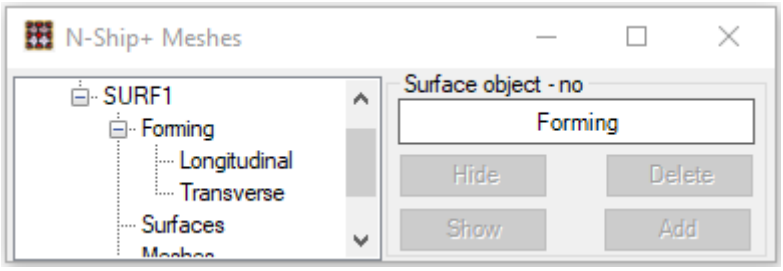


Figure 93. Shaping surface dialog box

In the right window of the Surfaces tab there is an inscription for the selected object in the tree: **Surface object** – no or – yes.

Attention: Shapeformers are subdivided into **Longitudinal** and **Transverse** for drawing surfaces by section (LOFT) and must be splines.

Limitation: The surface is formed by cross sections only.

All surface objects, if any, can be deleted, shown or removed using the corresponding buttons.

Attention: The **Delete** button does not delete objects from the model, but deletes them from the corresponding object in the surface tree.

When you select **Surfaces** in the surface tree and click the **Add** button in the right part of the window, the list of ways to create a surface, shown in Figure 94, is expanded.

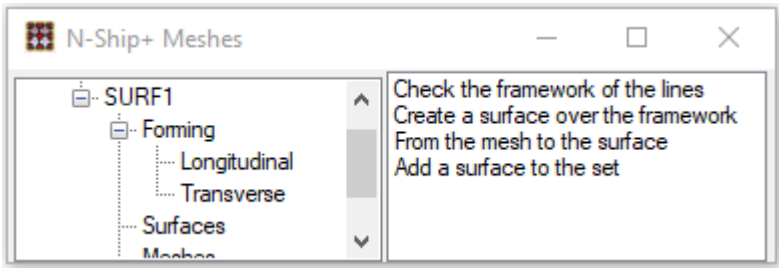


Figure 94. A dialog box with a list of surface creation methods

If you select **Check the framework of the lines** in the list, the procedure of checking for intersection of frame lines is started.

If you select **Create a surface over the framework** in the list, the procedure of drawing the drawing surface by sections is started.

If you select **From the mesh to the surface** in the list, the procedure of drawing surface by sections from polygonal mesh is started.

If you select **Add a surface to the set** in the list, the procedure of creating a group of surfaces is started.

When you select the **Meshes** item in the surface tree and click the **Add** button in the right part of the Surfaces tab, the list of ways to create a surface meshes, shown in Figure 95, is expanded.

Unfolding

Mesh of lines

Equidistanta

Transformation

Mesh segment

Cut the mesh

Unite meshes

Meshes

Pipe

Profile

Gofr

Figure 95. List of ways to create surface meshes

Exit from the list is performed by specifying any item in the surface tree.

If you select the line **Unfoldingin** the list, the procedure of forming a sweep of the specified mesh and placing it in the specified point of the model is started.

Selecting the line **Mesh of lines** in the list allows you to form a network from a set of lines with a specified number of points in N direction.

Attention: Lines are any lines and splines (they must not intersect each other between the start and end points).

At first a window with the request shown in Figure 96 pops up.

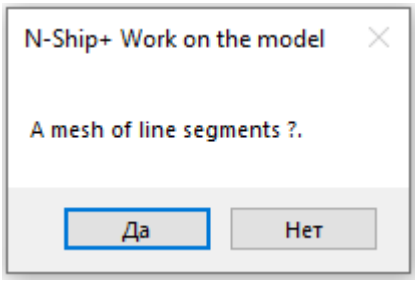


Figure 96. Request dialog box

When you click Yes on the command line, the prompt is displayed:

->Indicate the first boundary line of the property:

-> Indicate the second boundary line of the property:

-> Specify a set of forming lines:

The window shown in Figure 97 then pops up.

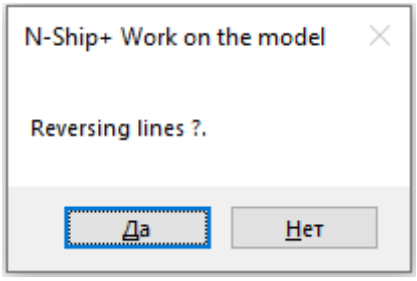


Figure 97. Request dialog box

Line directions (shown by arrows on the lines - the lines must be of the same direction, if necessary, you can change the direction of the indicated lines), then the command line prompts you:

->Specify the starting point of the mesh:

Then the number of approximation points to build the mesh is requested (not more than 255) and the network is built. The mesh is added to the named group of meshes.

If you select the **No** button, the procedure asks for a set of lines, then asks for the direction of the lines, then asks for the start line of the network and then asks for the number of approximation points to build the mesh and the mesh is built. The mesh is added to the named group of meshes.

Selecting the **Equidistanta** in the list starts the procedure of mesh formation at the specified distance and in the specified direction.

Selecting the **Transformation** in the list is used to change the direction of the regular mesh.

Selecting the **Mesh segment** in the list starts the procedure of selecting a section on the mesh.

Selecting the **Cut the mesh** in the list launches the procedure of cutting the mesh either by the track given by two points or by the line generated on the mesh beforehand.

Selecting the **Unite meshes** in the list starts the procedure of forming a single mesh on the set of meshes specified at the request in the command line. It is important to keep in mind that the dimensionality of the meshes on the edge to be joined must be the same.

Selecting the **Meshes** in the list is intended for entering an existing mesh into the group of meshes (all other methods enter meshes into the group automatically).

Selecting the **Pipe** in the list starts the procedure of network formation for the pipe.

When the **Profile** is selected in the list, the window looks as shown in Figure 98.

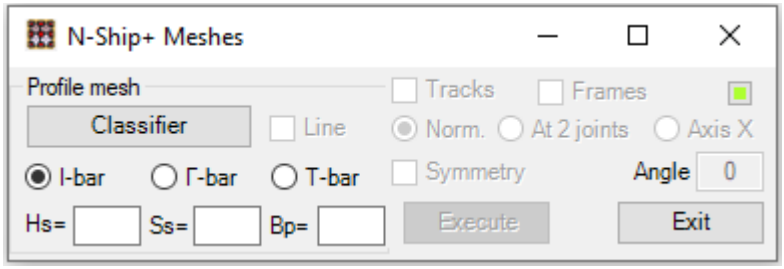


Figure 98. Dialog window with parameters for building the profile mesh

In this window it is necessary to set profile parameters, conditions of its installation in relation to the basic sections (Frames), a set of traces (longitudinal spatial lines), on which the meshes will be built.

When clicking on the **Classifier** button, the window will have the view shown in Figure 99.

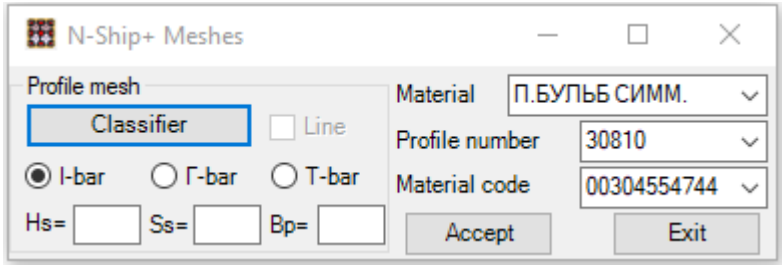


Figure 99. Dialog box when the button is clicked **Classifier**

In the **Material** drop-down list select the profile type, in the **Profile number** drop-down list select the profile number, and in the **Material code** drop-down list select the material code according to the material classifier.

By the **Accept** button the profile parameters will be transferred to the window in the left part. In this window it is necessary to activate the corresponding switch of the formed profile mesh or the **Line** pointer.

The profile mesh is set by switches:

I-shaped - only the wall mesh is built,

L-shaped or **T**-shaped - wall and shelf mesh is built.

When the **Line** pointer is activated, the mesh will be built along the section line of the profile.

In the input field **Hs** sets the height of the profile wall, in the input field **Ss** sets the thickness of the profile wall, and in the input field **Bp** sets the width of the shelf. All dimensions are set in mm.

If the shelf width is not specified, the mesh on the shelf is not built at any type of profile.

If a **T**-shaped profile is defined, the value of the wall thickness with a sign can be used to shift the flange mesh relative to the wall.

After setting the parameters for building the profile mesh, the **Tracks** pointer is unblocked, activation of which starts the procedure of selection of spatial trace lines in the model, then the **Frames** pointer is unblocked, activation of which starts the procedure of selection of base lines in the model, and the switches of the profile wall installation method are unblocked - **Norm**, **By 2 joints**, **Axis X** and the **Angle** input field.

Attention: By clicking on the green button next to the **Frames** pointer, the auxiliary procedure of highlighting traces and frames in the model is called.

When the **Norm.** switch is activated, the mesh formers are drawn normal to the base section at the intersection points of the section and the trace. If an angle is specified, the normal is rotated by the specified angle. If the **Axis X** switch is activated, the forming mesh are drawn at a specified angle to the X-axis of the world coordinate system. If you set the condition **At 2 joints**, first of all the edge (junction) formations are set. All intermediate mesh formations will be rotated so that the transition from one junction to another junction is smooth.

If the **Symmetry** pointer is activated, the profile mesh will be asymmetrized from the base axis.

If you press the **Execute** button, the procedure of forming wall, shelf or profile meshes is started.

If you select **Gofr** in the list, the procedure of forming the surface mesh by the pre-defined flat forming line of the gofr in the direction perpendicular to the plane of the gofr line is started..

When you select **Stretches** in the **Surface** tree and click the **Add** button on the right side of the **Surfaces** tab, the list of ways to create a surface stretch, shown in Figure 100, expands.

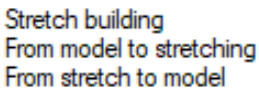


Figure 100. A list of ways to create a stretch

Exit the list by specifying any item in the surface tree.

If you select the **Stretch building** method, the procedure of stretching formation by the specified lines of zero buttock and the set of frame sections of the model is started.

If you select the **From model to stretch** method, the procedure of transferring the specified model lines to the stretch is started.

If you select **From stretch to model**, the procedure of transferring the specified stretch lines to the model is started.

5.5. Button Intersections

When you press the button  the “N-Ship+ Intersections” window pops up, which has the form shown in Figure 101.

Figure 101. Object intersection dialog box

The initial state of the window - everything is locked except **The set (1), The set (2), Add(1)**, **Points**, **Lines** and **Layer**.

Pressing **The set (2)** button will change the number in the button inscriptions to 2.

If you press **The set (1)** button, the number on the buttons will change to 1.

In the fields under the **The set** buttons the message about the presence of model primitives in the corresponding set is displayed.

If you press the **Add** button, the procedure of selecting objects from the model with the formation of the set of objects is started, about what in the corresponding field appears the inscription - **Primitive set**.

If you press the **Delete** button, the procedure of removing primitives from the set is started, you can delete all primitives or those specified in the model on request.

If you press the **Show** button, the procedure of highlighting primitives of the set is started.

The Intersections tab is intended for intersection of primitives of two sets with each other or intersection of primitives of one set with a plane, the way of definition of which is specified from the **Select** list.

If two sets are defined, the **Execute** button is unlocked and pressing it calls the procedure of intersection of primitives of the sets with each other.

If model objects are added to one of the sets, **Primitive set** appears, the **Select** list is unlocked, the **Execute** button is locked and the window will look as shown in Figure 102.

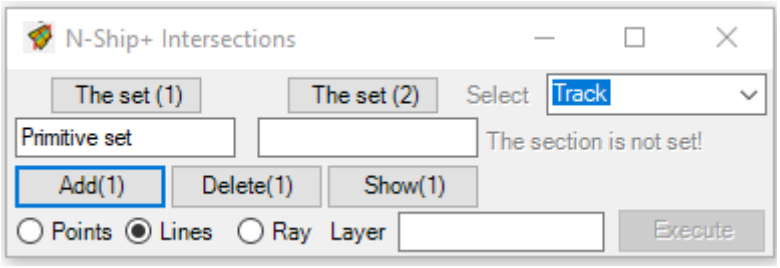


Figure 102. Dialog box after selecting model objects

To define a plane/planes of intersection with a set of primitives, you must select a plane definition from the **Select** list shown in Figure 103.

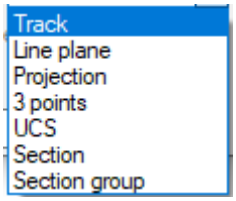


Figure 103. **Select** list

The plane can be set as **Track** through 2 points in the model, with the section plane perpendicular to the screen plane. You can set the plane as **Line plane**, which is selected in the model, and the line must be flat. You can set the plane as a **Projection** of a line in the model to the plane of the current UCS. The plane can be defined through **3 points** in the model. The plane can be defined as the plane of the named UCS in the model, from the list that is expanded by pointing to the **UCS** line.

It is possible to define the **Section** using the spacing table or by specifying a point in the model.

If you select the method of drawing the plane **Section**, the window will have the form shown in Figure 104.

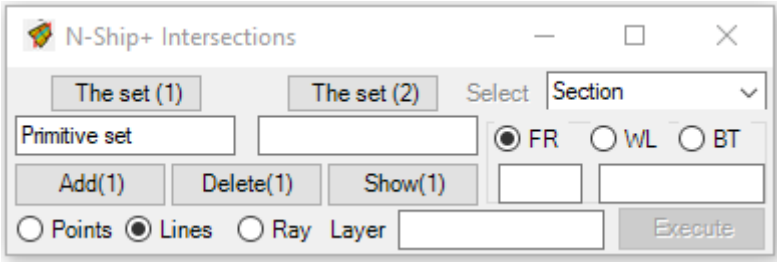


Figure 104. Dialog box for the **Section** method

It is necessary to set the switch **FR**, **WL** or **BT** and enter in the input fields the corresponding number and offset with a sign. After entering the offset and pressing the **Execute** button, the parameters of the corresponding plane will be displayed in the window.

Attention: You can use a point in the model to get the number and offset of the plane. The procedure of point selection is started by double click in the section number input field.

If you select the method of drawing the plane **Section group**, the window will look as shown in Figure 105.

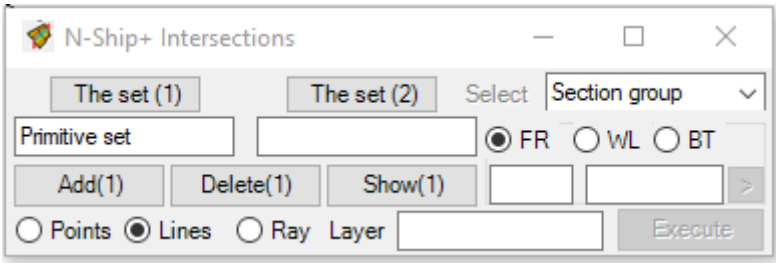


Figure 105. Dialog box for **Section group** method

To get the number and offset of the first section plane, you can use a point in the model or set the number and offset.

After entering the position of the first section, the button is unlocked  and by pressing it you can set the position of the second section for a group of sections, the button will change to  .

To get the number and offset of the second section plane, you can use a point in the model or set the number and offset. By clicking on the button  you can return to the first section setting.

After entering the position of the second cross-section it is necessary to place the cursor in the offset field and press the enter key. The window will look as shown in the figure 106.

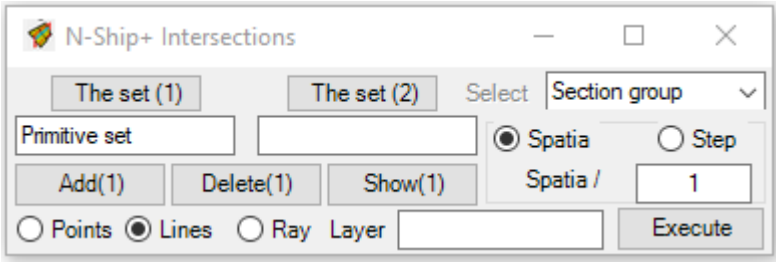


Figure 106. Dialog box for specifying sections

It is necessary to activate either the **Spatia** switch, if it is necessary to specify intermediate sections by dividing the spatia value by the value specified in the input field, or **Step**, specified in the input field.

When the **Execute** button is pressed, a message about section group definition is displayed in the window.

If the section/sections are defined, the **Execute** button will be unlocked and clicking on it will start the procedure of intersection of the set of primitives with the plane/sections.

Attention: When selecting objects in the model, you must respond to the NanoCAD command line. The procedures are looped to define several similar section planes at once.

The **Points**, **Lines** and **Ray** switches define the result of the intersection procedure.

In the **Layer** input field, you specify the name of the layer that is assigned to the result. When you point to the input field, the window shown in Figure 107 pops up.

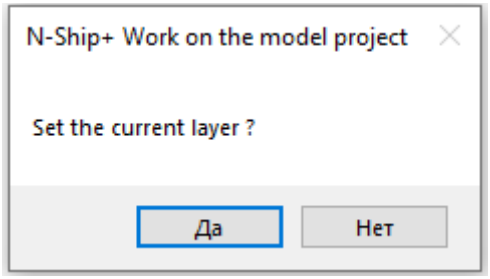


Figure 107. Request dialog box

If you answer **Yes**, the name of the current layer is transferred to the layer name field. In the layer input field you can enter new layers, it is desirable to form the layer name taking into account the rules of naming accepted in the N-Ship+ system.

The intersection result is defined by the switches - **Points**, **Lines** or **Ray**.

The intersection procedure is started by pressing the **Execute** button.

5.6. Working with model objects

When you expand the drawing item in the model tree, the function panel will have the appearance shown in Figure 108.

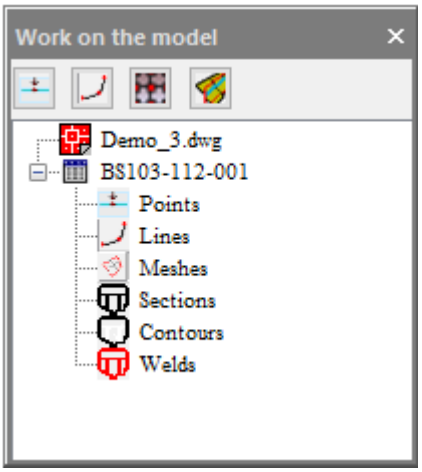


Figure 108. Function panel with drawing objects

The drawing context menu has the view shown in Figure 109.

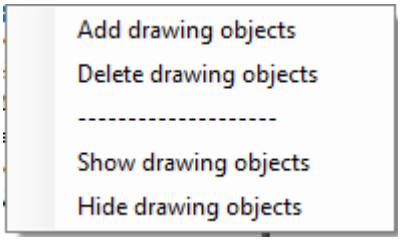


Figure 109. Drawing context menu

The context menu items are used to add, delete and visualize in the model groups of model objects represented under the drawing.

The context menu **Points**, is shown in Figure 110.

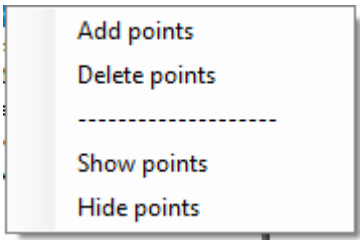


Figure 110. Points object context menu

The context menu items are used to add, delete and visualize in the model a group of points for a given drawing.

The context menu Lines, shown in Figure 111

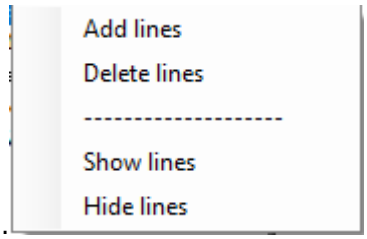


Figure 111. Context menu of the **Lines** object

The context menu items are used to add, delete and visualize in the model a group of lines for a given drawing.

The context menu of **Meshes**, is shown in Figure 112.

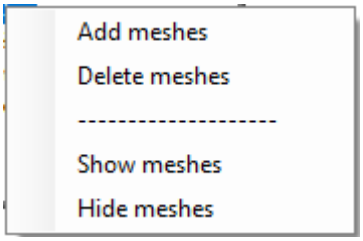


Figure 112. Context menu of the **Meshes** object

The context menu items are used to add, delete and visualize in the model a group of nets for a given drawing.

The context menu **Sections**, is shown in Figure 113.

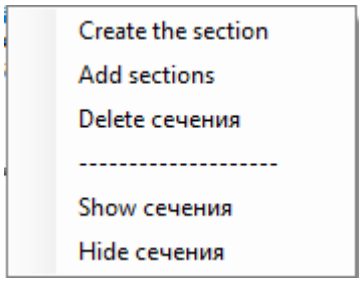


Figure 113. Section object context menu

The context menu items are used to add, delete and visualize in the model a group of plane sections for the given drawing.

Attention: The first item of the context menu Create section calls the procedure of creating a plane section in the model from the lines selected in the model. After creating the flat section lines, it is necessary to create contours and then add the lines to the section group, otherwise there will be some difficulties when creating contours.

The **Contours** context menu is shown in Figure 114.

The context menu items are used to add, delete and visualize in the model a group of flat closed contours for a given drawing. The Contours object is used to create flat sheet parts.

Attention: The first item of the context menu BOUNDARY calls the standard nanoCAD command.

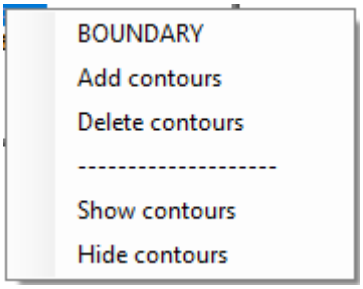


Figure 114. Contours object context menu

The **Welds** context menu, shown in Figure 115.

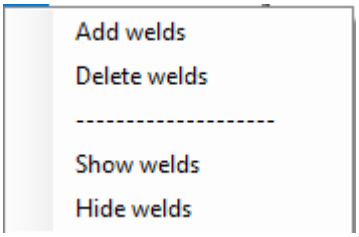


Figure 115. Welds object context menu

The **Welds** context menu items are used to add, delete and visualize in the model a group of welds for a given drawing.

6. WORKING WITH MODEL PARTS

6.1. Command window interface **MODEL PARTS ...**

The program complex of the **MODEL PARTS...** command allows you to generate in the model file environment:

- Solid of parts;
- flat workpieces of parts, including knices and brackets with flanges;
- flat workpieces (sweeps) of bent parts;
- flat workpieces of parts made of rolled sections;
- flat workpieces for belts;
- multiple parts.

The function panel called by the **Part of model** menu command is shown in Figure 116. This panel contains the specification tree window of the active order.

Attention: Before executing the menu command **ДЕТАЛИ МОДЕЛИ…** you must activate the project in the **BDATA** module.

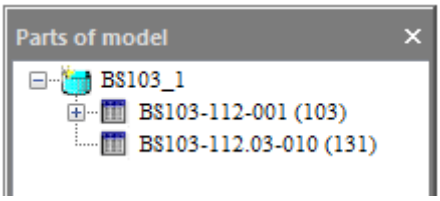


Figure 116. Function panel **Part of model**

If you click on the + next to the drawing, the drawing tree will look as shown in Figure 117.

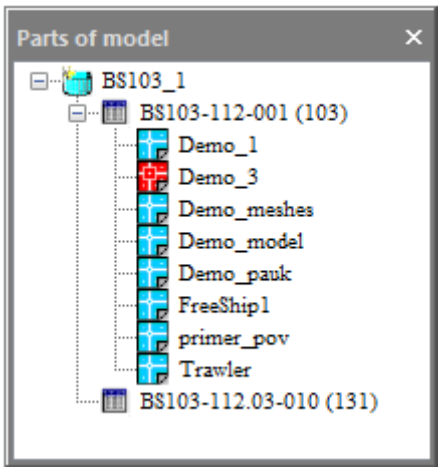


Figure 117. Function panel **Part of model**

Under the drawing, all models that were used to create the objects of the specified drawing will be shown, with the currently active model shown in red.

Each item in the specification tree of the active order, except for models, has a context menu that pops up when you select an object in the tree.

The context menu of the order object has the form shown in Figure 118.

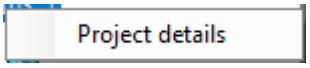


Figure 118. Project object context menu

When you click on **Project details**, the window shown in Figure 119 pops up.

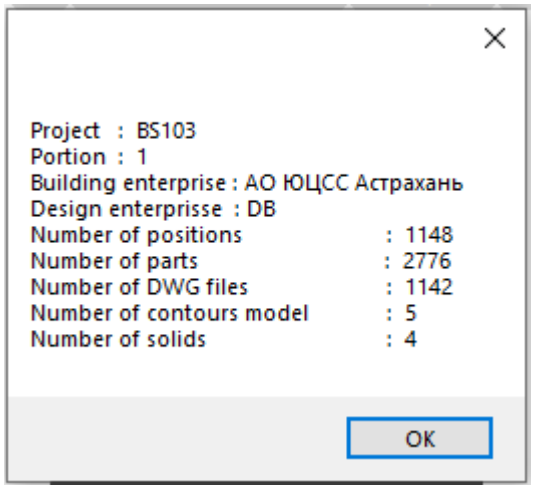


Figure 119. Project details window

The window shows the order details at the current moment.

The context menu of the drawing object has the form shown in Figure 120.

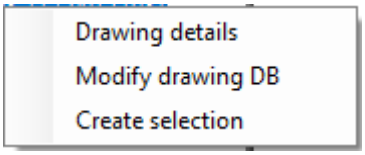


Figure 120. Drawing object context menu

When you click **Drawing details**, the window shown in Figure 121 pops up.

The window shows the current details of the specified drawing.

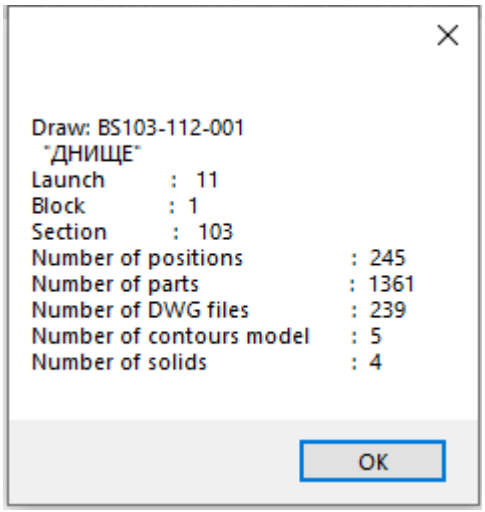


Figure 121. Drawing details window

When you click **Modify drawing DB**, the window shown in Figure 122 pops up.

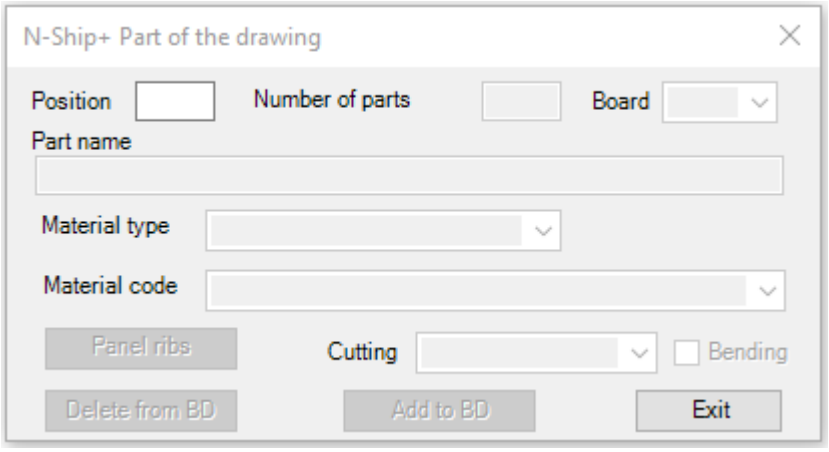


Figure 122. **Part of the drawing** dialog box

In this window all fields except the **Position** field are locked, exit the window by pressing the **Exit** button.

For a new part it is necessary to set successively **Position**, **Number of parts**, **Board**, **Part name**, **Material type** and **Material code** from the offered lists, the **Cutting** from the list and, if necessary, Bending. The lists **Material type** and **Material code** are formed according to the material classifier of the given drawing, if the required material is not available, it must be defined in **BDATA** module.

If you enter in the **Position** field the part number of the part available in the specification, the query shown in Figure 123 will pop up.

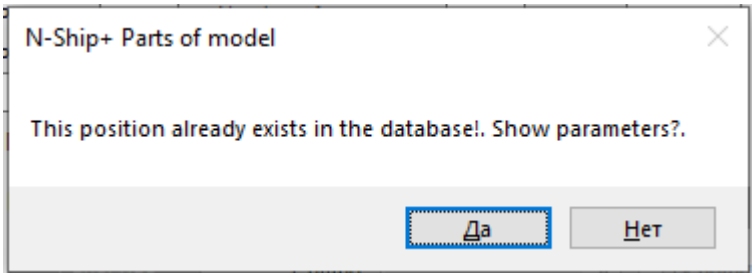


Figure 123. Request window

If you click on the Yes button, the window will be filled with the pattern parameters from the database and will look as shown in Figure 124.

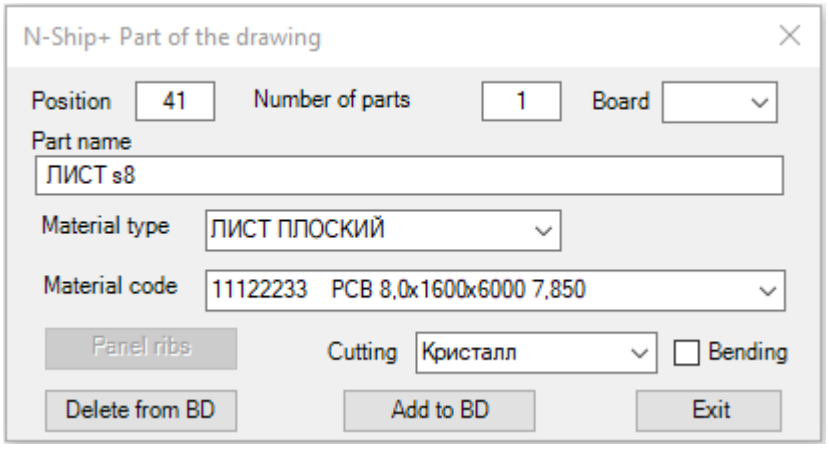


Figure 124. Part parameters dialog box

If the part material is PANEL the window will have the appearance shown in Figure 125.

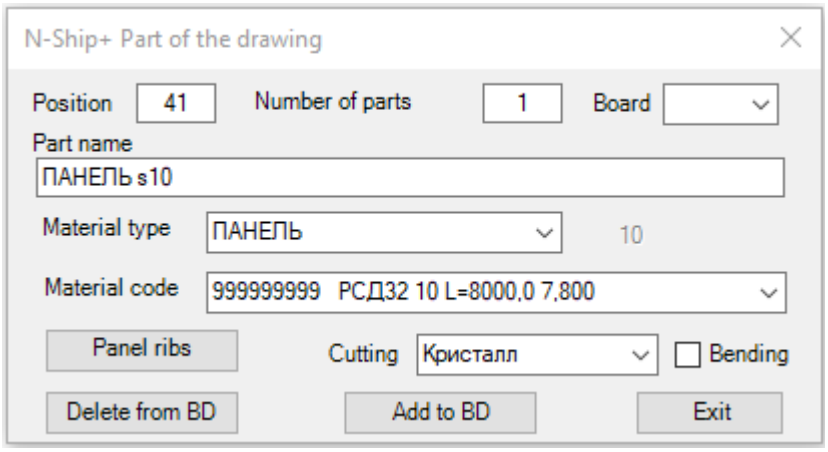


Figure 125. Part parameters dialog box from the panel

Attention: If the panel stiffeners are already set, the line with the material type will contain the number of the corresponding profile. If there is no profile number, it is necessary to click on **Panel ribs** button. If there are no profiles with the material brand of the panel in the material database, the window shown in Figure 126 will pop up.

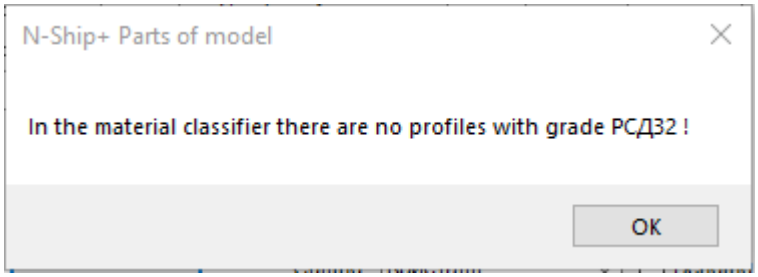


Figure 126. Window with the message about the absence of profiles with the specified grade

If there are profiles of the specified brand, the window shown in Figure 127 will pop up.

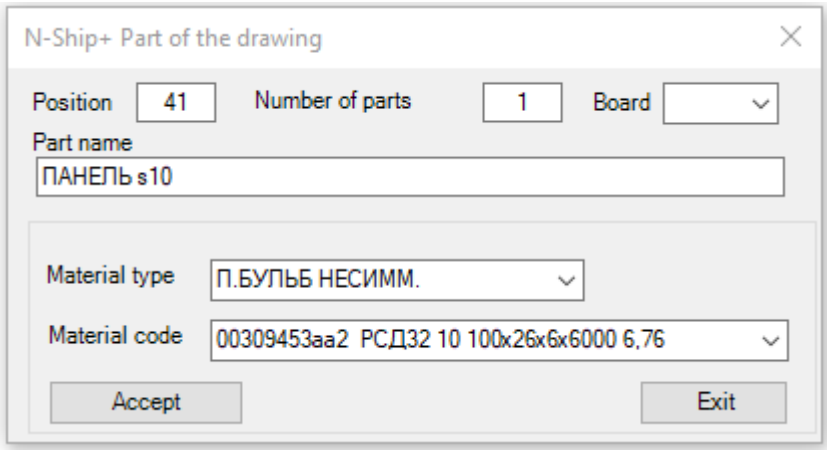


Figure 127. Dialog box with parameters of the panel's RL parameters

It is necessary to select the type and material code of the panel stiffeners and click on the **Accept** button, after which you will return to the window shown in Figure 125.

Exit the window for entering the panel stiffener material code and return to the previous window using the **Exit** button.

If you click on **Create selection** (Figure 120), the functional panel will have the view shown in Figure 128.

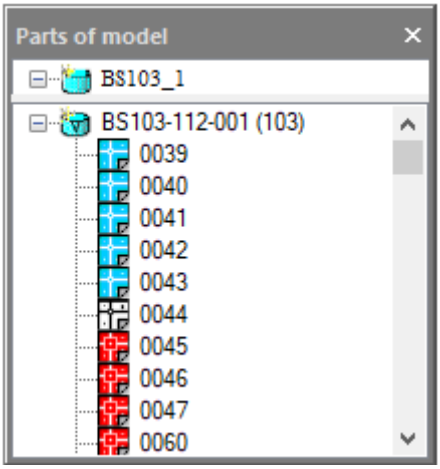


Figure 128. Function panel when clicking on **Create selection**

Attention: Initially the pattern has only attribute information in the database and is displayed in the tree with a white colored icon. After forming and writing the pattern in the MDET module environment, the icon color will be blue. After you have written the pattern in the PART module environment, the icon color will be red.

The context menu of the drawing object has the view shown in Figure 129.

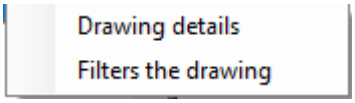


Figure 129. **Drawing details** context menu

When you click **Drawing details**, the window shown in Figure 130 pops up.

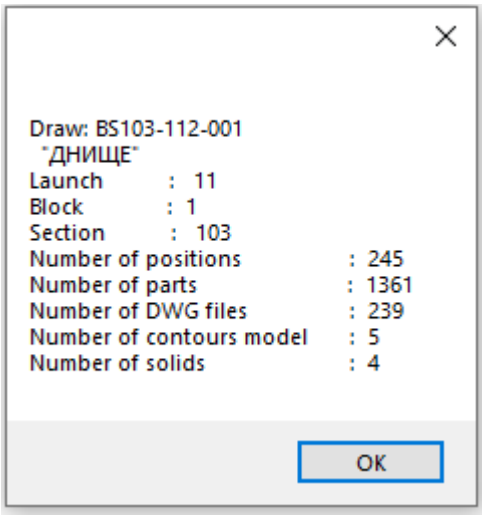


Figure 130. Window with drawing details

The window shows the current details of the specified drawing.

When you click on **Filter the drawing,** the window shown in Figure 131 pops up.

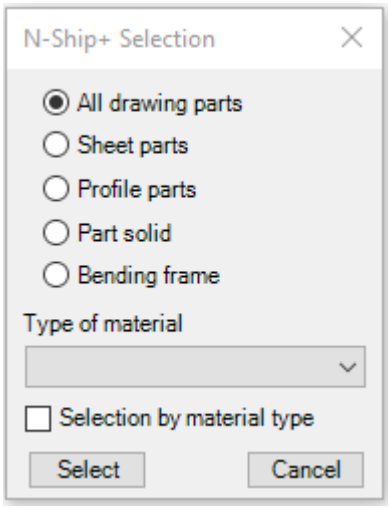


Figure 131. **Filter the drawing** dialog box

The window shows five switches, **Type of material** list, a **Selection by material type** pointer and the **Select** and **Cancel** buttons.

The active radio button shows which objects will be displayed in the parts tree of the drawing.

Selecting **Type of material** from the list and activating the **Selection by material type** pointer are intended to visualize in the tree only the parts with the specified material.

If you click on the **Select** button, the patterns will be selected according to the specified conditions and visualized in the drawing objects tree.

If you click on **Cancel** button, no selection is performed.

Attention: In order to remove the parts tree in the lower part of the panel, you must specify the project in the tree in the upper part of the function panel.

The context menu of the pattern object depends on the filling of the pattern data in the database.

Initially, the pattern has only attribute information in the database and is displayed in the tree with a white colored icon and the context menu is as shown in Figure 132.

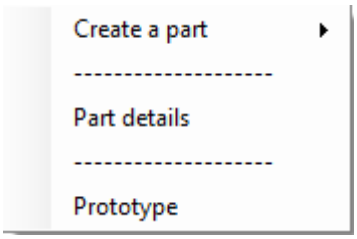


Figure 132. Context menu of the part object

Depending on the type of material from which the part is made, the context menu of the **Create a part** item will have a different design.

If the part is a sheet metal part of belt type, the context menu will have the form shown in Figure 133.

Figure 133. Context menu of the part object

If the part is of panel type, the context menu will have the form shown in Figure 134.

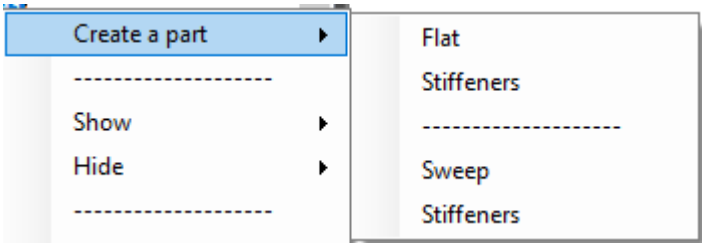


Figure 134. Context menu of the part object

If the part is a rolled section, the context menu will have the form shown in Figure 135.

Figure 135. Context menu of the part object

After creating a pattern by the **Create a part** procedure, the files of the flat blank and the pattern body are created in the database. The pattern is displayed in the tree with a red colored icon and the con-text menus of the **Show** and **Hide** items have the appearance shown in Figure 136.

Figure 136. Context menu of the part object

If you create a bent part reamer from a sheet and define the bending tool formation, the context menu of the Show and Hide items looks as shown in Figure 137.

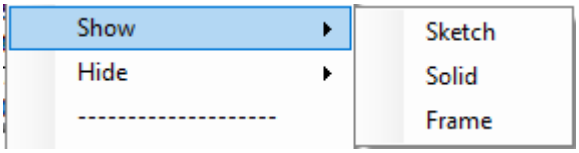


Figure 137. Context menu of the part object

If a template drawing is created from the bending frame, the context menu will have the form shown in Figure 138.

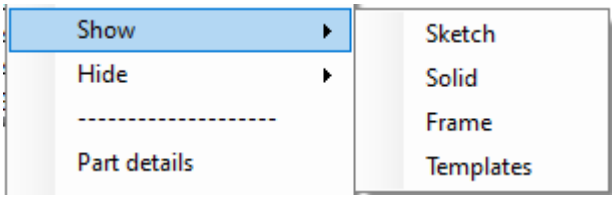


Figure 138. Context menu of the part object

When selecting **Show**, the procedure of reading the corresponding file from the database is started.

When selecting **Hide**, if the file is open, it will be closed.

To create a part using an existing prototype part, there is **Prototype** item in the context menu. Before selecting this item, you must call the prototype pattern body to the model and set it to the position

of the pattern to be created using nanoCAD tools (move, mirror, etc.). When you select the **Prototype** item, you will be prompted:

-> Specify the part block:

You need to specify the part block and then on request:

-> Symmetry draft?

If necessary, you can specify that the pattern sketch should be symmetrized.

If the part is displayed in the tree with a blue colored icon, when you select the context menu item **Drawing details**, the window shown in Figure 139 will pop up.

If the part is displayed in the tree with a blue color icon, which means that the pattern has a body, or a red color icon, which means that the pattern is technologically enclosed, the window shown in Figure 140 will appear.

The window shows the current details of the specified drawing part.

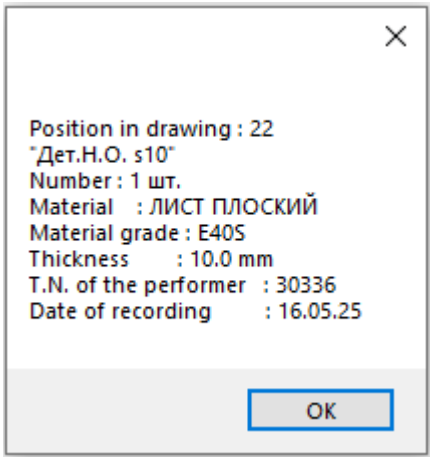


Figure 139. Window with drawing details

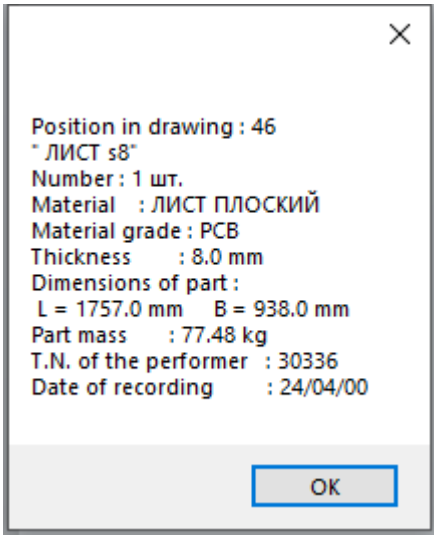


Figure 140. Window with drawing details

If you select the **Create part->Flat** context menu item, the window shown in Figure 142 will pop up.

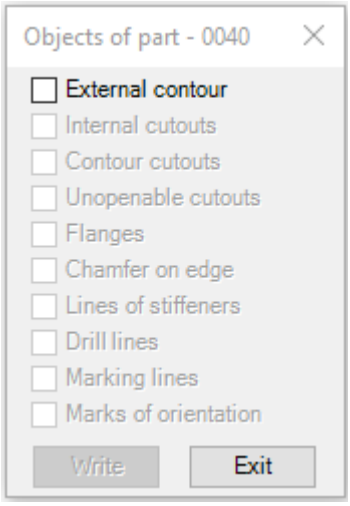


Figure 141. Window with Obiect of part

If you click on the **External contour** pointer, the procedure of drawing a pattern from a closedcontour on layer 0 as the base object of the model starts.

If you activate the **Create Part->Sweep** pointer, the window shown in Figure 142 will pop up.

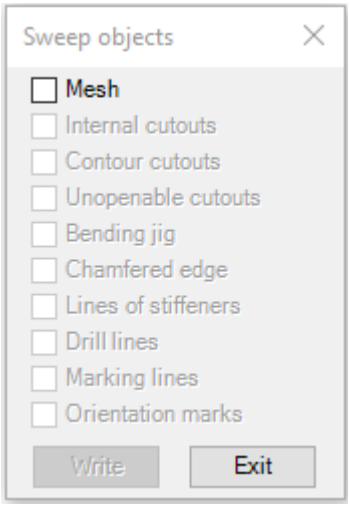


Figure 142. Window with Sweep objects

При нажатии на указатель **Сеть** запускается процедура построения детали из полигональной сети в качестве базового объекта модели.

When you select the context menu item **Create a part->Belt**, if the pattern is made of PROFILE ROW material, the window shown in Figure 143 will pop up.

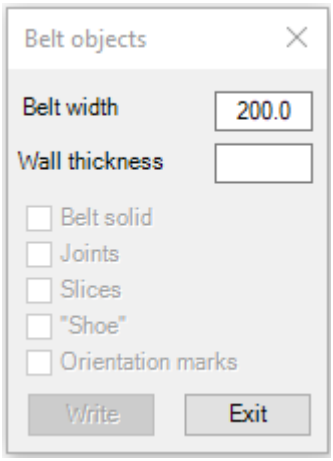


Figure 143. Window with belt objects

In this window in the **Belt width** input field the belt width is specified. In the **Wall thickness** input field it is necessary to specify the thickness of the wall supported by the belt, after that the **Belt solid** pointer will be unlocked.

Clicking on the **Belt solid** pointer will start the belt construction procedure.

If you activate the **Create part->Belt** for a part from flat, the window shown in Figure 144 will pop up.

In this window, in the **Belt width** input field, you must enter the belt width and press the **Enter** key.

In the unlocked input field **Wall thickness** you must enter the thickness of the wall to be supported by the belt and press the **Enter** key.

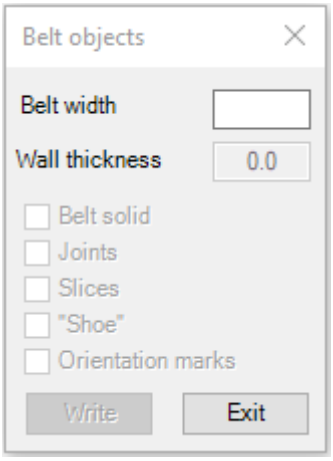


Figure 144. Window with belt objects

Clicking on the **Belt solid** pointer will start the belt construction procedure.

If you select **Create part->Profile** from the context menu, the window shown in Figure 145 will pop up.

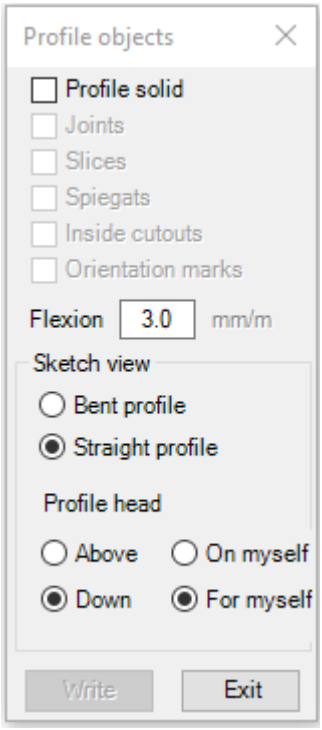


Рисунок 145. Диалоговое окно **Объекты профиля**

If you click on the **Profile solid** pointer, the procedure of building a part from profile rolled products is started.

If there are objects in the drawing object tree after filtering, if the **Part solid** switch is active (Figure 131), which indicates that there are pattern bodies in the database, then the drawing pattern tree will have the form shown in Figure 146.

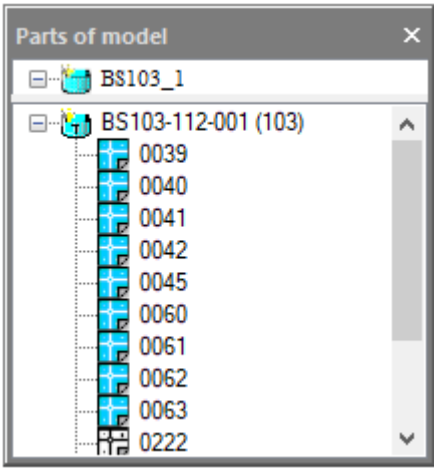


Figure 146. Function panel **Part of model**

The white color icon means that the number of these solids in the drawing is greater than one and the solid is one and you must add blocks.

To visualize the pattern bodies in the model, you can create a group of patterns in a tree. In the context menu of the drawing item, shown in Figure 147, you must select the **Create selection** item.

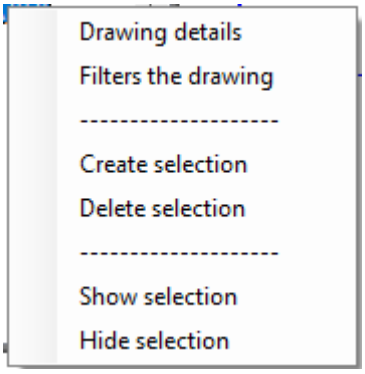


Figure 147. Context menu of the drawing item

The window shown in Figure 148 pops up.

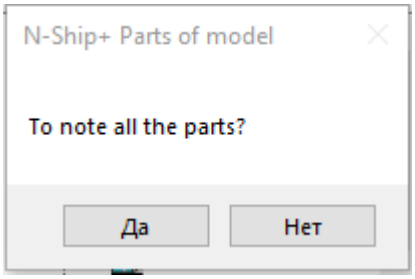


Figure 148. Request to form a group

If you click Yes, all the patterns will be checked, if you click No, the function panel will look as shown in Figure 149. The check marks in the white boxes will be used to mark the parts that will be included in the group.

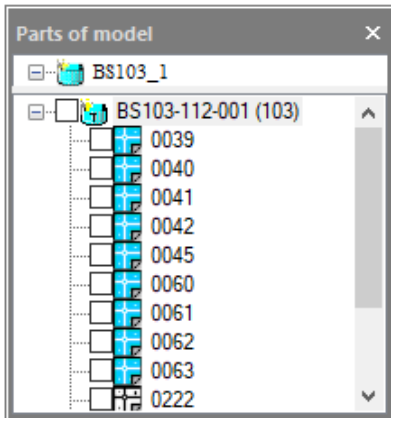


Figure 149. Function panel Part of model

If you select the **Delete selection** context menu item, the functional panel of the part becomes the same as it was before.

The selection is used to show in the model a group of drawing parts bodies when you select **Show** and **Hide** from the context menu.

The context menu of a drawing pattern is shown in Figure 150.

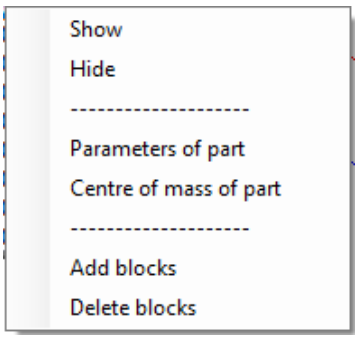


Figure 150. Drawing part context menu

The **Show** and **Hide** context menu items are used to visualize the body of the part.

When you select **Parameters of part**, the window shown in Figure 151 appears.

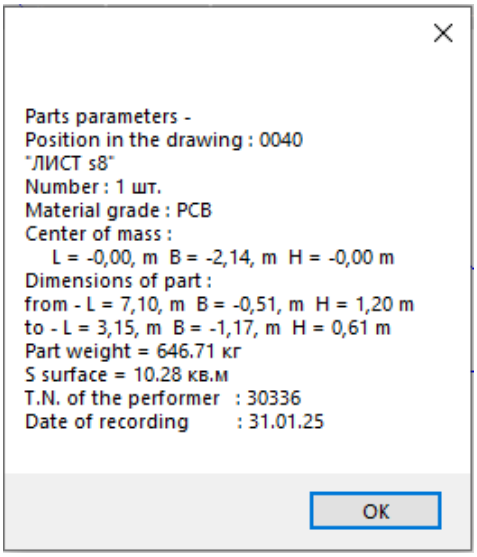


Figure 151. Window with part parameters

If you select **Center of mass of part**, you will call the procedure of reading the parameters of the pattern center of mass from the database and drawing its position in the model as a red point.

If in the database for a position in the drawing the quantity parameter is greater than one, you can perform a procedure to clone this pattern if necessary. To do this, you must install the part in the model environment and copy by means of NanoCADa the block of this part so many times that the total number of blocks is equal to the number of parts. The copies must be placed in the model according to the drawing. Then in the context menu you must select **Add blocks** to call the procedure of forming a set of pattern blocks and writing these blocks to the database, the specification tree will be rebuilt, under the corresponding position the pattern numbers will be formed with an index starting from one. Each copy of the pattern can be set in the model environment.

If in the drawing objects tree, after filtering, if the **Frame** switch is active (Figure 131), there are objects that indicate the presence in the database of the corresponding bending frames for bent sheet parts, the context menu of these objects has the form shown in Figure 152.

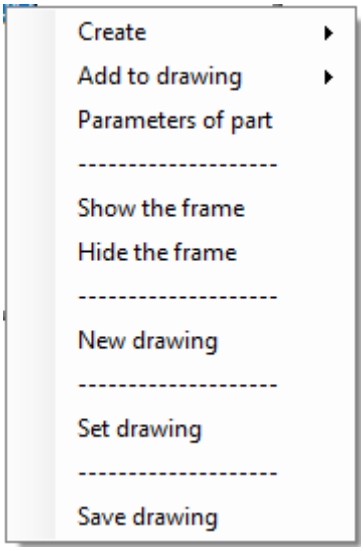


Figure 152. Context menu of objects of type bending frame

Selecting **Parameters of part** context menu item brings up the window shown in Figure 139.

If you select the context menu item **Show the frame**, the procedure of reading and setting the bending frame model on the screen is called.

If you select the context menu item **Hide the frame**, the procedure of closing the bending frame model is called.

Forming the drawing of the bending frame templates is performed in two steps. At the first stage, after setting the bending frame model, the blocks **General view**, **Scheme** and **Templates** are formed using the **Create** context menu item. At the second stage, after forming a new drawing by the context menu item **New drawing** with the required number of sheets, by the item **Add to drawing**, the **General view**, **Scheme** and **Templates** blocks are set on the drawing sheets of the frame templates.

6.2. Creating a flat part

If you activate the **Create a part ->Flat** pointer, you will be able to create a flat part from sheet metal in the window shown in Figure 153.

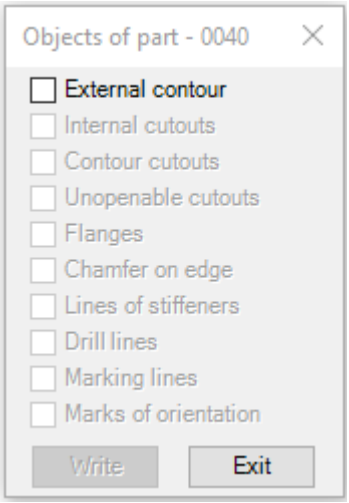


Figure 153. Objects of part dialog box

In this window all pointers are locked except for the **External contour**.

Exit from the **Objects of part** window by clicking on the **Exit** button.

If you activate the **External contour** pointer, you will execute the procedure to select in the model the external contour of the flat part.

Attention: The external contour of a part is a closed flat polyline on layer 0. It can be obtained by using the **BOUNDARY** command on the flat section of the model.

After defining the external contour, the pointers in the **Objects of part** window and the **Write** button will be unlocked.

If you activate the **Internal cutouts** pointer, you will start the procedure of selecting the internal cuts of the part in the model.

If you activate the **Contour cutouts** pointer, you will start the procedure of selecting the contour cuts of the pattern in the model.

If you activate the **Unopenable cutouts** pointer, you will start the procedure of selecting in the model the contour cutouts of the pattern.

Activating the **Flange** pointer opens the dialog box shown in Figure 154.

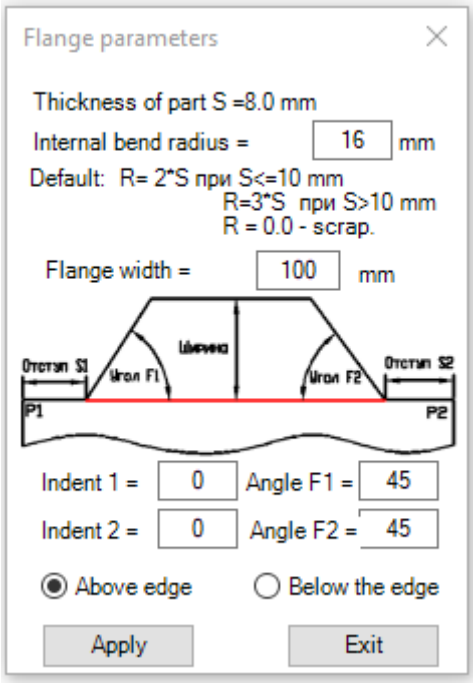


Figure 154. **Flange parameters** dialog box

In this window you must set the **Flange Width**, **Indent 1**, **Angle F1**, **Indent 2**, **Angle F2**, activate the **Above Edge** or **Below the edge** switch and click on **Apply** button, which starts the flange forming procedure on the edge of the pattern.

Attention: The **Below the edge** switch must not be used for neighboring patterns edges, because the result will be an overlap between the neighboring flanges.

Exit the window by clicking on the **Exit** without flange formation button.

Activating the **Chamfer on edge** pointer starts the procedure of chamfering on the part edge, the window shown in Figure 155 will pop up.

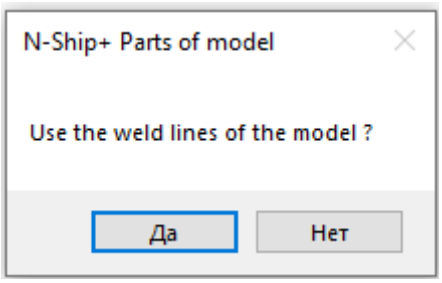


Figure 155. Request dialog box

If you click Yes, you will be prompted at the command line

-> Specify the weld in the model или [?]:

Attention: Welds in the model can be formed in the function panel called by the menu command **WORK ON MODEL.....**

If you click on the No button, the window shown in Figure 156 will pop up.

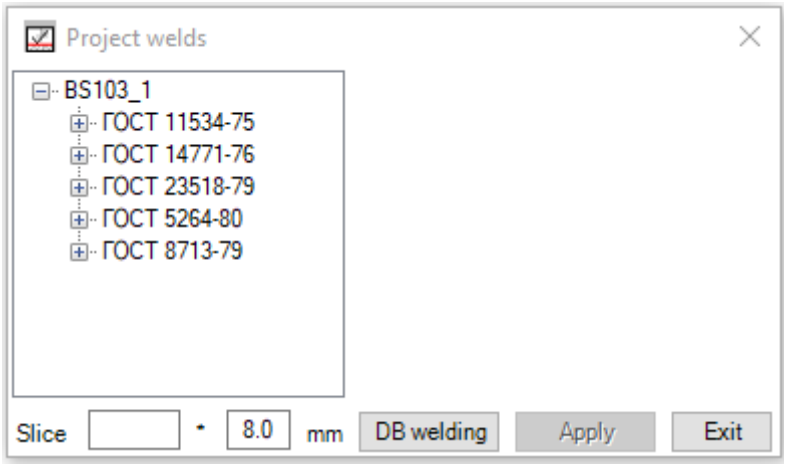


Figure 156. **Project welds** dialog box

Attention: The bottom line shows the thickness of the part to make it easier to define the fret. To set the weasel you must enter in the first field the removal width and in the second field the cut value.

In the tree of GOSTs for welds it is necessary to specify the type or kind of weld and click on the Apply button, which starts the procedure of chamfering on the edge of the part. During the procedure it is necessary to answer the queries in the command line of NanoCAD.

Attention: N-Ship+ system comes with a database of welds for the most used Gosts for welds. There are five standards parameters: GOST 8713-79, GOST 5264-80, GOST 14771-76, GOST 11534-75 and GOST 23518-79. Parameters of welds specified in the standards are copied to the database of the current project from the catalog of the N-Ship+ system at the first call of the command **DETAILS MODEL....**.

When you click on the button **DB welding**, the window shown in Figure 157 pops up.

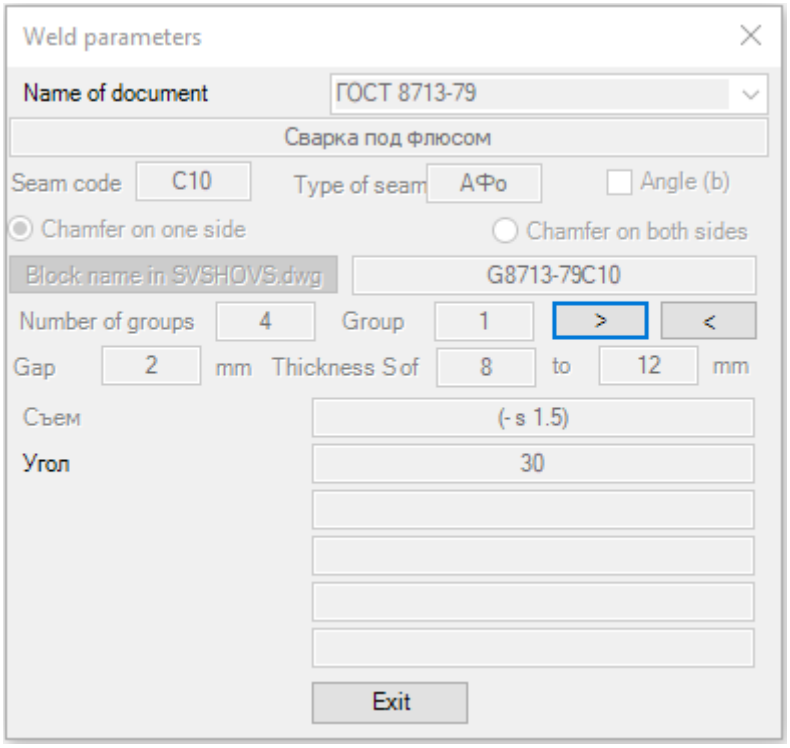


Figure 157. **Weld parameters** dialog box

In the **Weld parameters** window you can view the parameters of the weld.

Attention: Entering new documents, welds or correcting weld parameters in the project database is described in section 8 “WORKING WITH THE PROJECT WELD DATABASE”.

Activation of the **Lines of stiffeners** pointer starts the procedure of selecting the joining lines for the stiffeners on the flat section of the model.

Activating the **Drill lines** pointer starts the procedure of selecting a drill line model on a flat section.

Activating the **Marking lines** pointer starts the procedure for selecting marking lines on the flat section of the model.

Activating the **Marks of orientation** pointer starts the procedure of placing orientation marks on the part, from the window shown in Figure 158.

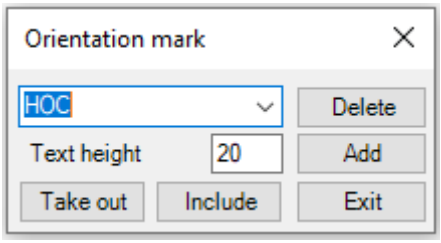


Figure 158. **Orientation mark** dialog box

This window contains the list of orientation text marks, the **Text height** input field and the function buttons - **Delete**, **Add**, **Take out**, **Include** and **Exit**.

The **Delete** and **Add** buttons are intended for corresponding operations with the list.

The **Take out** and **Include** buttons are used to insert and remove the selected orientation mark on the part field.

6.3. Create a flat sweep of a sheet part.

If you activate the **Create a part->Sweep** pointer, the creation of the sheet metal sweep of part will be done in the window shown in Figure 159.

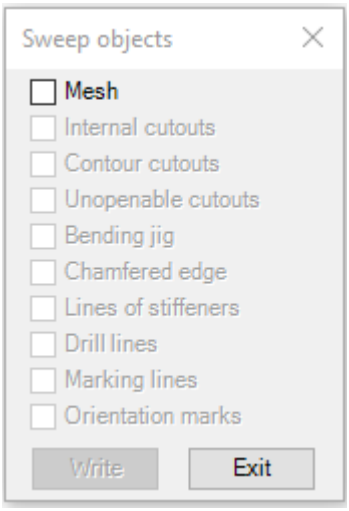


Figure 159. **Sweep objects** dialog box

When you activate the **Mesh** pointer, you must specify as base object the mesh to be generated by means of the functional tab **WORK ON MODEL...** .

Attention: Parts that have a break must be defined with a mesh and break lines.

After specifying a net in the model, you are prompted to orient the part thickness relative to the net surface.

If you activate the **Internal cutouts**, **Contour cutouts** or **Unopenable cutouts** pointers, the corresponding procedures will be started.

Attention: The cut lines must lie on the mesh, which is ensured by means of the functional tab **WORK ON MODEL...** .

When you activate the **Bending jig** pointer, the window shown in Figure 160 will pop up.

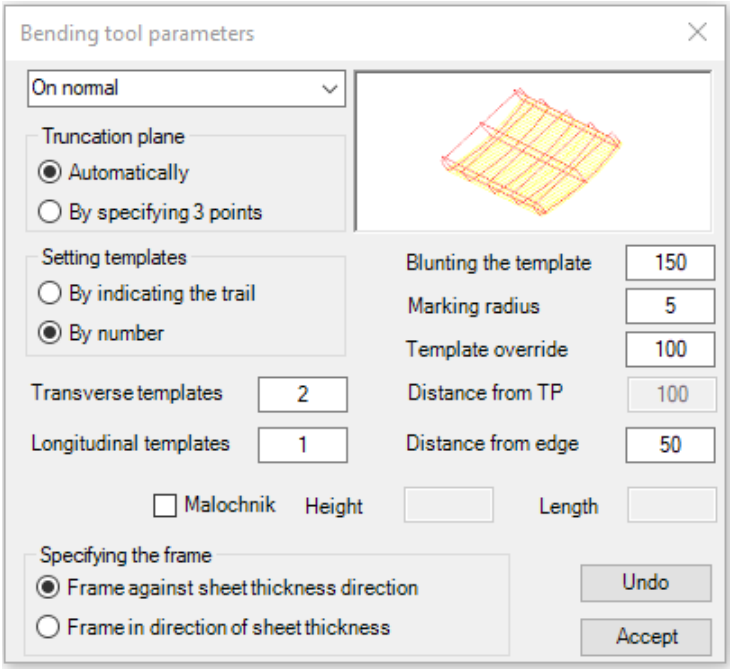


Figure 160. **Bending tool parameters** dialog box

Bending frame templates can be generated in three ways, which are defined in the Bending Tool Parameters drop-down list shown in Figure 161.

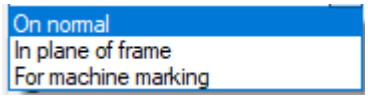


Figure 161. List of ways to generate bending frame templates

The **Normal** method is intended for generating templates in a plane normal to the truncation plane of the frame. Method **In plane of frame** is intended for generating templates in the plane, as a rule spandrels, and in this case the templates to the plane of truncation are installed under the malka.

Method **For machine marking** is designed to generate templates in the plane, usually frames, but in a special form for subsequent cutting and drawing on drafting machines with program control (it is necessary to set the **Distance from TP**).

The view of the defined bending frame is shown in in Figure 158.

The truncation plane of the frame can be defined **Automatically** or **By specifying 3 points** in the model. The position of the bending frame is set using the switches **Frame against sheet thickness direction** or **Frame in direction of sheet thickness** in the “Specifying the frame” window.

Attention: The direction of the sheet thickness in relation to the theoretical mesh is set when selecting the mesh.

You can set the position of the templates by specifying the trace of the intersection of the templates with the pattern surface by two points, or by specifying the line of the frame. You can set the number of transverse and longitudinal templates and the distance of the outermost templates from the sheet edge.

You can set the minimum **Blunting the template**, **Marking radius** for marking the points of intersection of the sheet edge with the templates and **Template override** of the from the sheet edge.

Activation of the **Malochnik** pointer starts the calculation of the angle of inclination of the templates with respect to the normal to the surface of the sheet part at the midpoint of the template cross-section. The angle calculates a small pattern in the form of a rectangle with height and length specified in the corresponding input fields (default 100x200 mm).

If you click on the **Accept** button, the set parameters will be memorized and the calculation of the bending frame will be performed when writing the sweep pattern.

If you click **Undo** button, the bending frame calculation will not be performed.

If you activate the **Lines of stiffeners**, **Drill lines** or **Marking lines** pointers, the corresponding procedures will be started.

If you activate the **Orientation marks** pointer, you will start the procedure to set the orientation marks on the pattern, from the window shown in Figure 158.

If you click on the **Write** button, the procedure of writing the pattern to the database is called.

6.4. Creation of a part from rolled sections.

Creation of a part from a rolled profile is performed in the window shown in Figure 162.

All pointers except the **Profile solid** are locked.

Setting of the view in the sketch - **Straight profile**, **Profile head** (**Down** and **For myself**).

Activation of the **Profile solid** pointer calls the procedure of profile body formation.

The procedure outputs a request to the command line

-> Specify connection line/mesh:

to define and calculate the contour of the part from the profile.

After defining the profile body, all pointers and the **Write** button are unlocked.

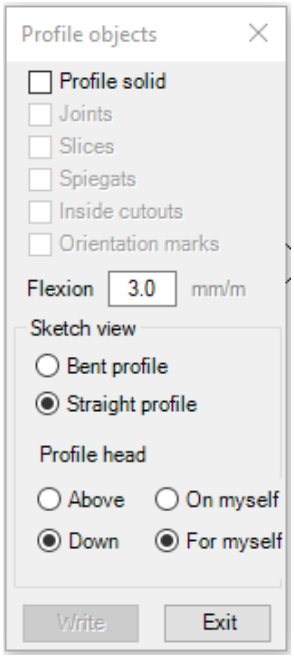


Figure 162. **Profile objects** dialogue box

Activating objects of the **Joints** type calls the procedure of defining the joints of the profile part.

Activating objects of the **Slices** type brings up the window shown in Figure 163.

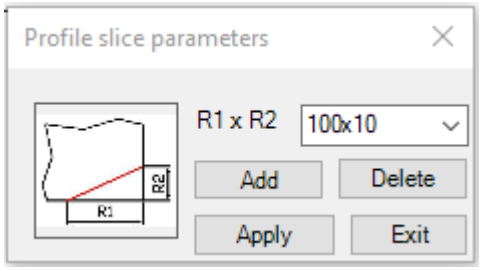


Figure 163. Profile slices parameters dialogue window

In the **Profile slice parameters** window in the **R1xR2** drop-down list select the line with cut parameters and click the **Apply** button. The procedure of slice construction will start.

The **R1xR2** list can be added and edited using the **Add** and **Delete** buttons.

Exit the window using the **Exit** button.

Activation of objects of the **Spiegats** type causes the window shown in Figure 164.

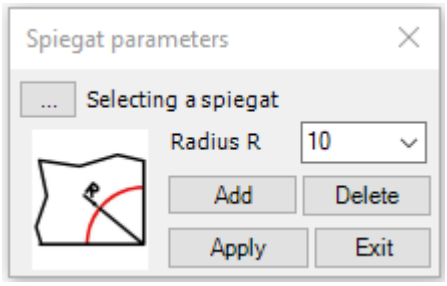


Figure 164. Spiegat parameters dialogue box

In the **Spiegat parameters** window in the **Radius R** drop-down list select the line with the parameters of the spiegat in the form of an arc with a specified radius to start the procedure of building the spiegat. The **Radius R** list can be added and edited using the **Add** and **Delete** buttons.

If you want to build a spiegat defined as a line, you should click on the **Select a spiegat** button and in the window shown in Figure 165 you should select a dwg file of the spiegat contour and click on the **Open** button.

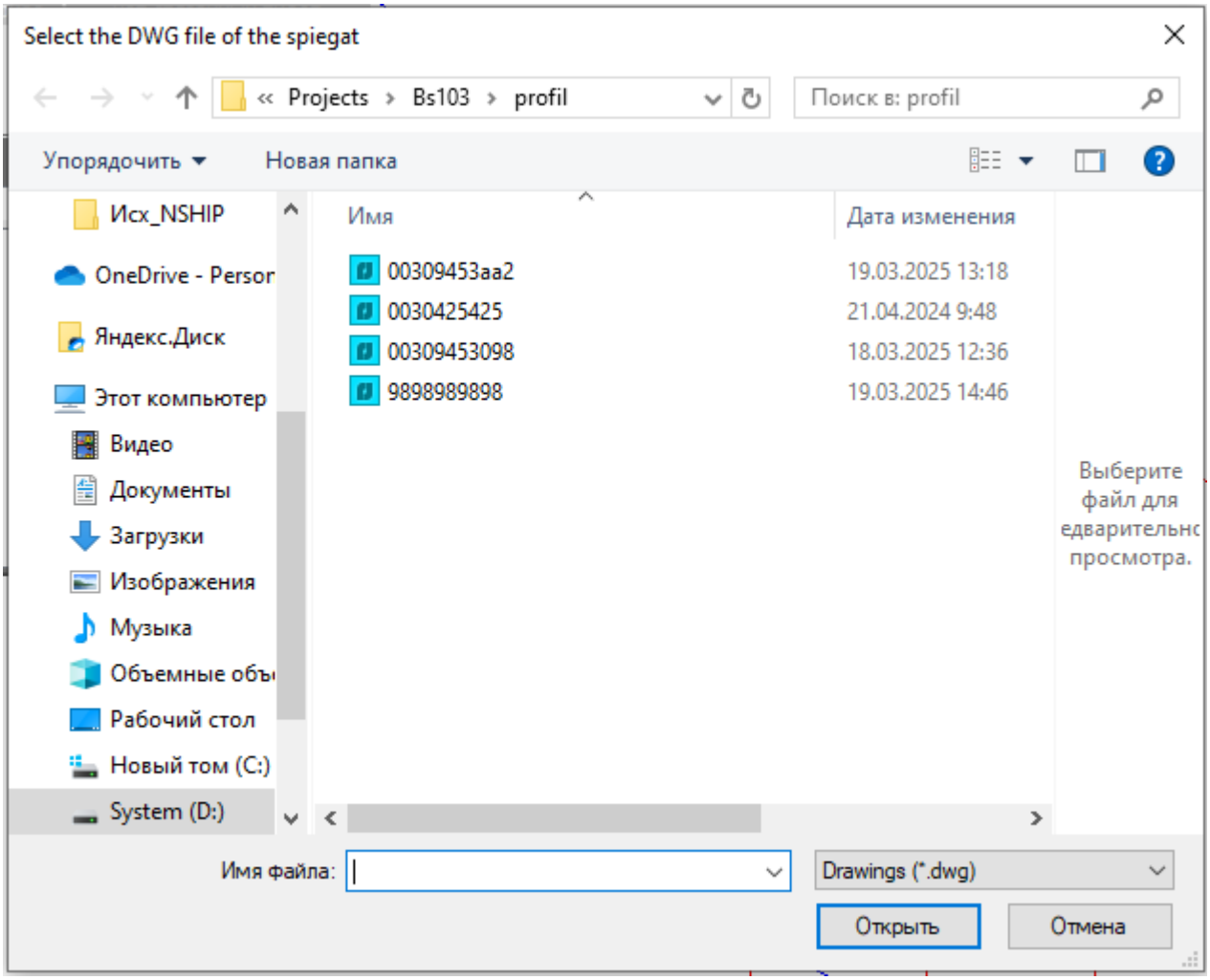


Figure 165. Dwg file selection dialogue box

Start the **Spiegat** procedure building by clicking the **Apply** button.

Exit the window by clicking the **Exit** button.

Activating objects of the **Inside cutouts** brings up the window shown in Figure 166.

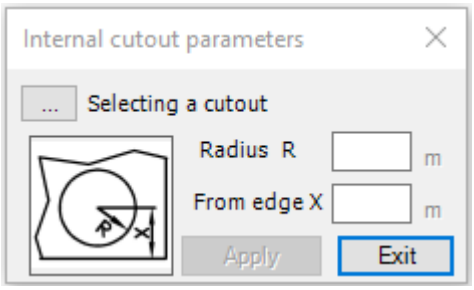


Figure 166. **Internal cutout parameters** dialogue box

The **Select a cutout** button is similar to the button for the spiegat.

The **Radius R** parameter must be set for a circular cutout.

Parameter **From edge X** sets the position of the cutout from the line of the profile joining.

Click on **Apply** button, which starts the procedure building internal cutout.

Exit the window by clicking the **Exit** button.

Activation of the **Orientation marks** pointer brings up the window shown in Figure 153.

6.5. Creation of a belt from sheet metal.

If you activate the **Create pattern->Belt** pointer, the creation of a belt pattern from sheet metal will be done in the window shown in Figure 167.

In the input field Belt width you must enter the belt width, if the material of the belt pattern is set as strip, then in this field you must enter the width of the strip from the database,

In the input field Wall thickness you must specify the thickness of the wall that is supported by the belt.

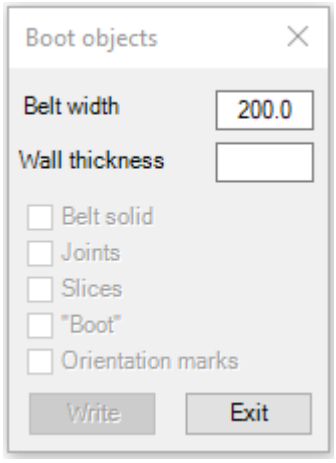


Figure 167. Belt objects dialogue box

If you activate the **Belt solid** pointer, you will be prompted to generate the belt body by asking for the model base line (the line of the pattern edge to which the belt adjoins at an angle of 90 degrees to the pattern plane). During the calculation, the thickness orientation and the displacement of the girdle relative to the base pattern plane are requested.

Once the girdle body is defined, the **Joints**, **Slices**, **"Boot"** and **Orientation marks** pointers are enabled.

Activating the **Joints** pointer will start the procedure of forming the joints of the belt.

Activating the **Slices** pointer brings up the window shown in Figure 163.

Activating the **Orientation Mark** pointer brings up the window shown in Figure 158.

Activating the **"Boot"** pointer opens the window shown in Figure 168.

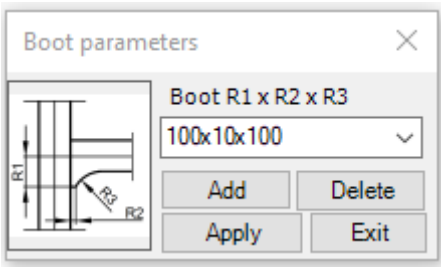


Figure 168. Boot parameters dialog box

In the **Boot parameters** window in the Boot R1xR2xR3 drop-down list, select the line with the parameters of the cobbler and click the **Apply** button. The procedure of bootstrap construction will start.

You can add and edit the Boot R1xR2xR3 list using the **Add** and **Delete** buttons.

Exit the window using the **Exit** button.

6.6. Creating a panel with stiffeners.

If the part material is defined as a panel with stiffeners (material code - 90), the context menu shown in Figure 169 pops up.

Figure 169. Context menu for panel type part

Activating the **Flat** pointer calls the procedure to generate a flat part, and activating the **Sweep** pointer calls the procedure to generate the part sweep, both procedures described above.

Attention: The panel **Stiffeners** pointers are activated only if there is a part solid created by the corresponding procedure.

When you activate the **Stiffeners** pointer of a flat part, the window shown in Figure 170 will pop up. The same window will pop up for a part sweep.

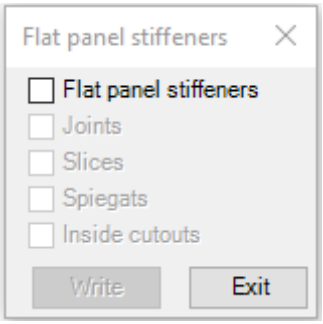


Figure 170. Flat panel stiffeners dialog box

Activation of the **Flat panel stiffeners** pointer starts the procedure of building the flat panel stiffeners

Activation of the **Joints** pointer starts the procedure of building joints on the stiffener panel.

Activation of the **Slices** pointer starts the procedure of drawing cuts on the stiffener panel.

Activation of the **Spigates** pointer starts the procedure of building scuppers on the stiffener panel.

Activation of the **Inside cutouts** pointer starts the procedure of creating cutouts on the stiffener. panel.

The same procedure is performed for the bent parts for which you have executed the sweep drawing procedure.

Attention: If the panel has the same stiffeners, it is enough to build one stiffener and then, using nanoCAD tools, copy this stiffener to the necessary positions on the panel.

The **Write** button is used to form the panel solid with stiffeners and exit the window.

The **Exit** button exits the window without saving the constructions.

6.7. Create a drawing of the bending frame templates

If a bending frame was formed during the formation by the sweep procedure and it is necessary to create a drawing of the templates of this frame, you must sort the drawing by the **Bending frame** switch (Figure 131).

The context menu of the pake in the sorted list is shown in Figure 171.

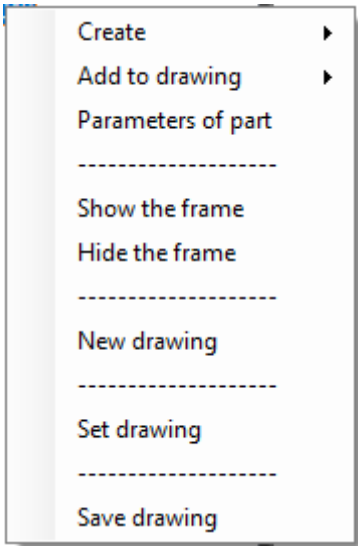


Figure 171. Context menu for part

At first it is necessary to install the bending frame model using the **Show the frame** item, if this is not done, then when clicking on any other item a window with a corresponding warning pops up.

The context menu of the Create command is shown in Figure 172.

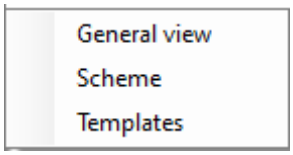


Figure 172. Context menu for Create

When selecting the **Create -> General View** context menu item, the procedure of general view block formation is started. In the process of the procedure execution the screen displays a query:

->Rotate view?

If **Yes** is entered, the free view rotation command is invoked to set the position at which the templates and the inscriptions indicating the templates are visible in the general view.

You are then prompted:

->Specify a rectangle of general view:

You need to specify two points of a rectangle covering a general view.

This is followed by a query:

->Redisplay the rectangular section of the general view ?.

If you enter **Yes**, the previous prompt will be issued again. to reset the rectangle in case not all common view objects fall within the previous rectangle.

Attention: The dimensions of the rectangle will be scaled when the common view block is placed on the drawing sheet, so it is necessary to set the rectangle as close as possible to the common view objects.

After creating the common view block the message shown in Figure 173 is displayed.

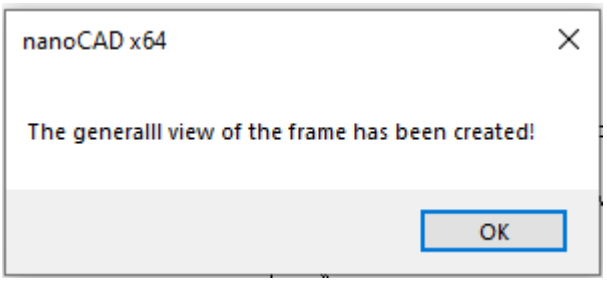


Figure 173. Message after the procedure has been completed

Attention: The created block with the name OVID can be seen in the Insert block window called in nanoCAD.

The **Create -> Scheme** context menu item starts the procedure of building a template installation scheme and displays a prompt in the command line:

->Specify the starting of the control line :

The point should be specified based on the fact that it is the beginning of the coordinate system for marking the scheme.

Next, the command line prompts you to enter a query:

->Specify the direction of the control line :

After that, a reference line with the CL inscription is drawn on the screen. The height of the text of the subsequent inscriptions will be equal to the height of the CL inscription, the height can be changed in the next query:-

>Reset the text heigth ?.:

If you specify **Yes**, you can set a new text height.

Next, you are prompted:

->Indicate the location of the caption :

t is necessary to specify a point in the middle of the caption to place the text “Template Setup Scheme”.

A request is then issued:

->Specify a rectangle of the view:

You need to specify two points of a rectangle encompassing a general view.

This is followed by a query:

->Redisplay the view rectangle ?.

If you enter **Yes**, the previous prompt will be issued again. to reset the right-angle in case not all of the schematic objects fall within the previous rectangle.

After the schematic block is created, the message shown in Figure 174 is displayed.

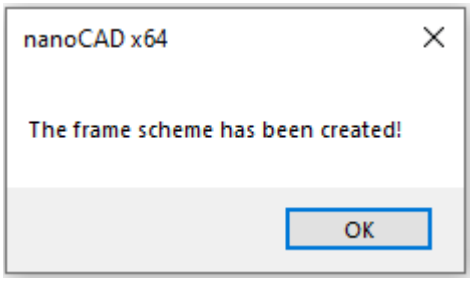


Figure 174. Message after the procedure has been completed

Attention: The created block with the name SXEMA can be seen in the Insert block window, called in nanoCAD.

When selecting the **Create ->Templates** context menu, the procedure of template blocks formation is started. The first template is formed on the screen and a query is displayed:

>Reset the text heigth ?.:

The height of the inscriptions can be judged by the inscription below the template, if the height needs to be changed, you should answer **Yes** to the query and set a new text height.

Then the query is issued

:->Укажите шаг <предлагаемый по умолчанию>:

The step should be set based on the convenience of marking when making templates manually.

Further on the request

->Replace step ?

You can set a different step value if you are not satisfied with the position of the inscriptions, etc.

A query is then issued:

->Specify a rectangle of the view:

You need to specify two points of a rectangle encompassing a general view.

This is followed by a query:

->Redisplay the view rectangle ?.

If you enter **Yes**, the previous prompt is reissued. to reset the right-angle in case not all of the pattern objects fall within the previous rectangle.

The process then continues for all bending frame templates.

After the pattern blocks are created, the message shown in Figure 175 is displayed.

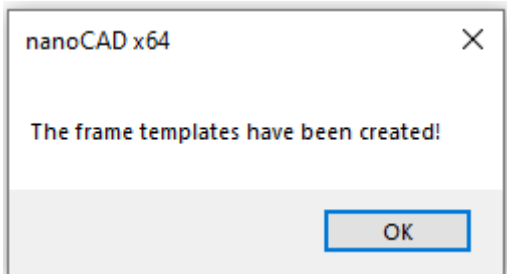


Figure 175. Message after the procedure has been completed

Attention: Created blocks with the name SHABL* can be seen in the Insert block window called in nanoCAD.

When you select the **New drawing** context menu item, a window pops up that looks like the one shown in Figure 176.

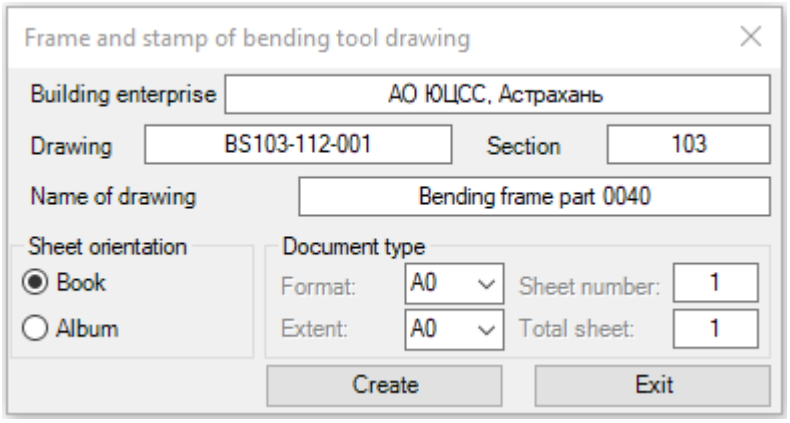


Figure 176. New drawing dialog box

In this window it is necessary to set the **Format**, **Extent**, **Sheet number**, **Total sheet**, as well as the layout of the drawing - **Book** or **Album**.

Attention: For bending tool drawings the number of sheets should be determined based on the number of templates.

If you press the **Create** button, the procedure of forming the drawing blank with a stamp will be called, exit without forming the drawing by pressing the **Exit** button.

After the drawing sheets are created, the mode of working with the first sheet is set on the screen.

When selecting the context menu item **Add to drawing** -> **General view**, the procedure of placing the General view block on the sheet is started.

At the request:

->Specify a rectangle of the view:

it is necessary to set a rectangle for placing the general view block on the sheet.

Similarly, the selected drawing sheet is filled with the Template layout block.

After the Template block is placed on the drawing sheet, the prompt shown in Figure 177 pops up.

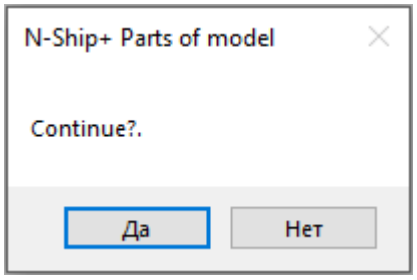


Figure 177. Request to continue the procedure

If the **No** button is pressed, the procedure is terminated and the warning message shown in Figure 178 is displayed.

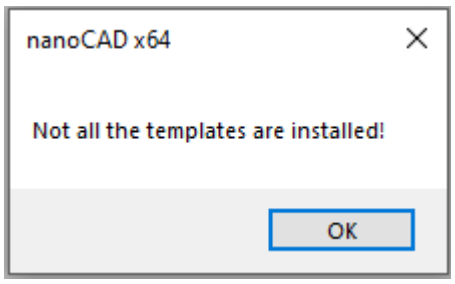


Figure 178. Warning message

Now you can specify the next drawing sheet to be filled with the next templates and repeat the procedure by specifying the context menu item Add to drawing -> Template.

After the last template is installed, the message shown in Figure 179 is displayed.

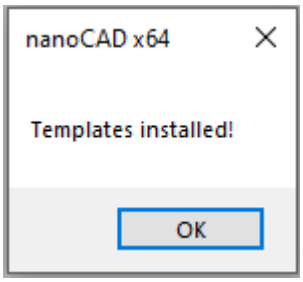


Figure 179. Warning message

Since the objects installed in the drawing are blocks, they can be moved and edited using nano-CAD tools.

If you select the **Save drawing** context menu item, the drawing will be saved to the database.

If the context menu item **Set drawing** is selected, the file from the database will be called to the screen.

7. CREATING TOOLING FOR THE SECTION

7.1. The window interface of the TOOL...

Figure 180 shows the “N-Ship+ Work with Project Models” dialog box, invoked by the command **TOOL...**

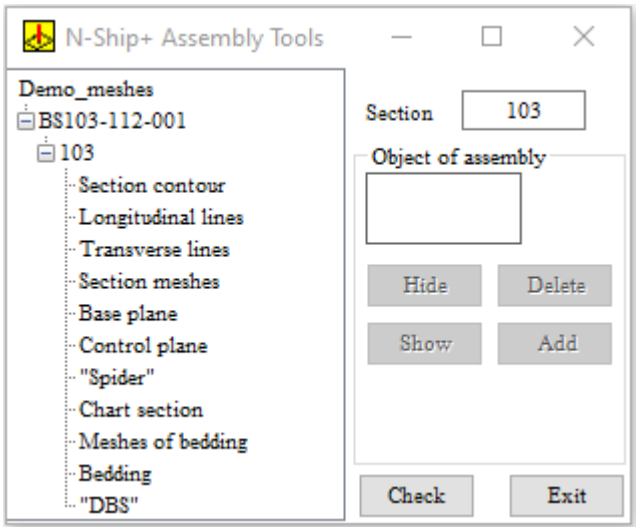


Figure 180. Dialog box Assembly Tools

The “N-Ship+ Assembly Tools” window contains a model tree window with drawing, section and fitting objects, a “Object of assembly” window and a number of function keys. The model tree window contains the name of the model file at the root, the drawing associated with the model and the section underneath it.

The window can be enlarged by specifying an edge or a corner of the window and stretching it.

Standard objects are presented under the section :

- - contour of section;
- - longitudinal lines of the frame;
- - transverse lines of the frame;
- - section meshes;
- - base plane;
- - control plane;
- - layout diagram of the base spar (spider);
- - layout diagram of longitudinal and transverse section sets (schematic);
- - bed meshes;
- - data for section bedding construction (bedding);
- - data for section coke bed setting (DBS).

To calculate the corresponding objects it is necessary to specify the object in the tree, and in the “Tooling object” window the presence of this object in the model is reflected - to the right of the object name - ‘Yes’ or “No” and the corresponding buttons **Hide**, **Delete**, **Show**, **Add** and **Parameters** (for objects that need them) are highlighted.

If the object does not exist, only the **Add** button is available, which, when clicked, starts the procedure for creating the object. As a rule, the procedure of creating an object makes a request to enter information from the command line. It is necessary to enter the corresponding information and the procedure creates an object, at the next pointing to which in the tree, to the right of the object name in the window “Object of assembly” appears the inscription “Yes” and the buttons **Hide**, **Delete** and **Show** are highlighted. The **Hide** and **Show** buttons start the corresponding procedures of highlighting the object primitives, and the **Delete** button starts the procedure of removing the object primitives from the model.

To check for the intersection of longitudinal and transverse lines representing the surface frame, when calculating the layout for marking the longitudinal and transverse set of the section, if necessary, you can run the line intersection procedure by clicking on the **Check** button.

If the lines cross each other, the window shown in Figure 181 will pop up.

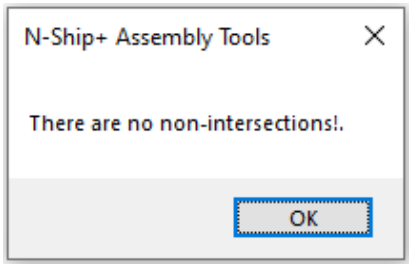


Figure 181. Message window

If there are non-intersections, the window shown in Figure 182 pops up, showing the number of non-intersections, and the lines that do not intersect are highlighted on the screen.

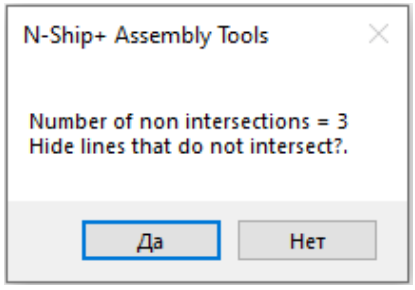


Figure 182. Message window

If you click **Yes**, the lines that do not intersect will be highlighted.

The **Base plane** object is the plane onto which the base frame layout and marking scheme will be projected, and it is also the base of the bed.

If you specify a **Base plane** object in the tree, if the object is already created, the window will have the appearance shown in Figure 183.

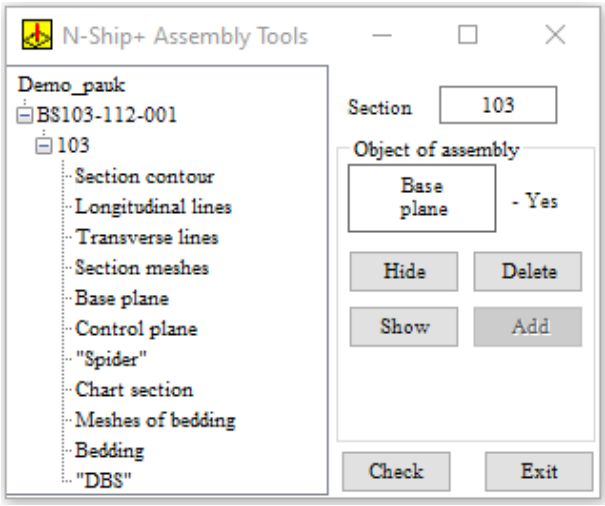


Figure 183. Dialog box Assembly Tools

If there is no **Base plane** object, when you click the Add button, the window shown in Figure 184 pops up.

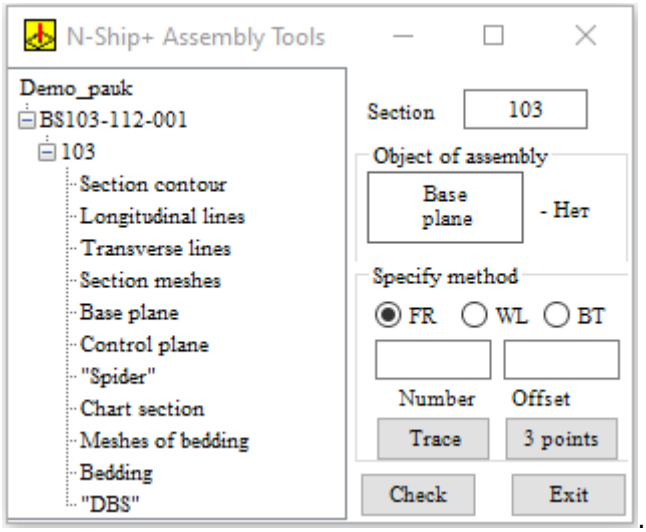


Figure 184. Dialog box Assembly Tools

In this window it is necessary to select the method of base plane formation. If you select the method using the project spacing table, you must activate the corresponding switch - FR, WL or BT and enter in the input fields the values of section number and offset from the specified section with a sign. If the specified position is outside the limits of the spacing table, a corresponding warning will be displayed (the construction will be performed within the limits of the spacing table).

You can set the reference plane perpendicular to the screen plane by specifying a trace through two points and you can set the model in the viewpoint and specify 3 predefined points in the model that define the reference plane.

The **Control plane** object is the plane of the reference line for Spider and schematic, and for Bedding it is the plane from which you draw the bed supports or the dimensions for the patterns.

For objects of Spider and Schematic type, the section meshes are specified as basic objects, these are the meshes on which the joining lines of the set are punched.

For the objects of the type Bed and DBS as basic objects are specified the meshes of bed, these are the meshes that are built taking into account the thickness of sheets N.O.

The objects of the type Spider, Schematic, Bed and DBS require for creation of parameters input. If objects of Spider or Scheme type are created, the Service button pops up in the “N-Ship+ Assembly Tools” window, as shown in Figure 185.

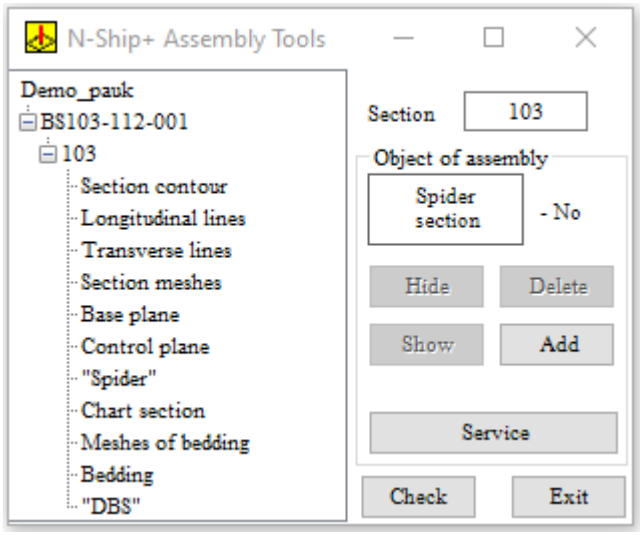


Figure 185. Assembly Tools dialog box

When you click the **Service** button, the N-Ship+ Assembly Tools window is rearranged as shown in Figure 186.

You can set the text height for the labels.

Clicking on the **Diagonal size** button starts the procedure of forming an arrow from the first point to the second point and generating the size text.

If you click on the S**tretch size** button, you start the procedure of forming a dimension link from the first set point of a given line to the next set points with generation of dimensions along the line.

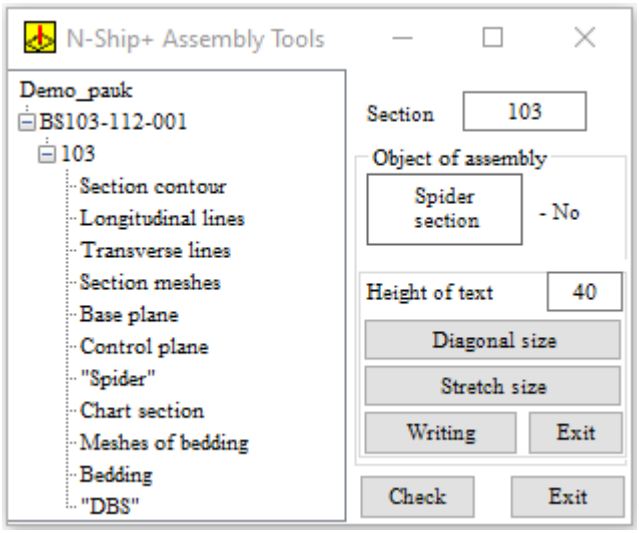


Figure 186. Assembly Tools dialog box

If you click the **Writing** button, the procedure of text line formation is started, if necessary.

Exit from the service by clicking the **Exit** button.

If an object of the **Bedding** type is created, when you click the **Parameters** button (in place of the Service button), the “N-Ship+ Assembly Tools” window is rebuilt as shown in Figure 187.

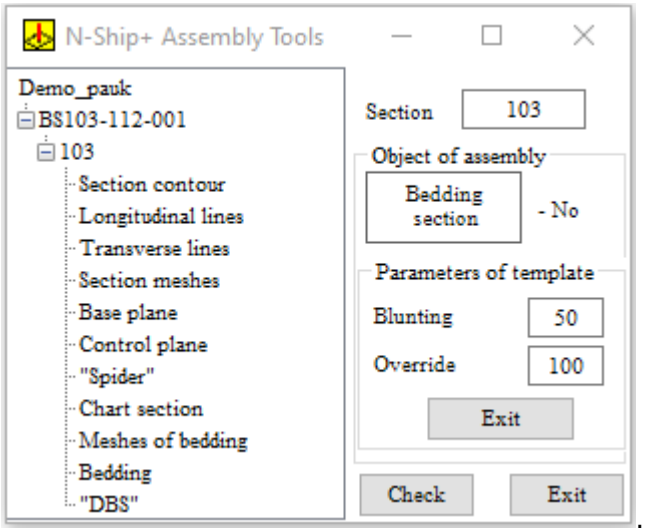


Figure 187. Assembly Tools dialog box

In this window it is possible to set the **Blunting** of the mold and **Override** of the mold over the edge of the skin and click the **Exit** button.

If you create objects of **DBS** type, then when you click the **Parameters** button, the “N-Ship+ Assembly Tools” window will be rearranged as shown in Figure 188.

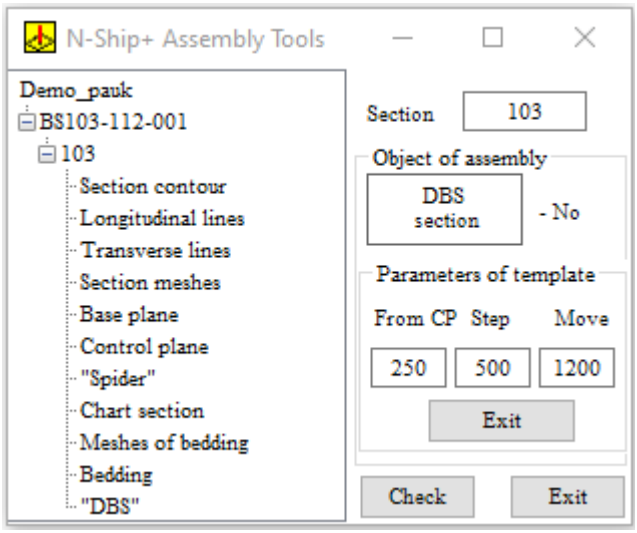


Figure 198. Assembly Tools dialog box

In this window you can set the distance of the first girder from the reference plane, the strut spacing and the maximum extension of the girder struts and press the **Exit** key.

Exit the “N-Ship+ Assembly Tools” window by pressing the **Exit** button.

8. WORKING WITH THE PROJECT WELD DATABASE

8.1. Command **WELDING SEAMS...** of the **MDET** module

The **WELDING SEAMS...** command calls the “Project welds” window, shown in Figure 199.

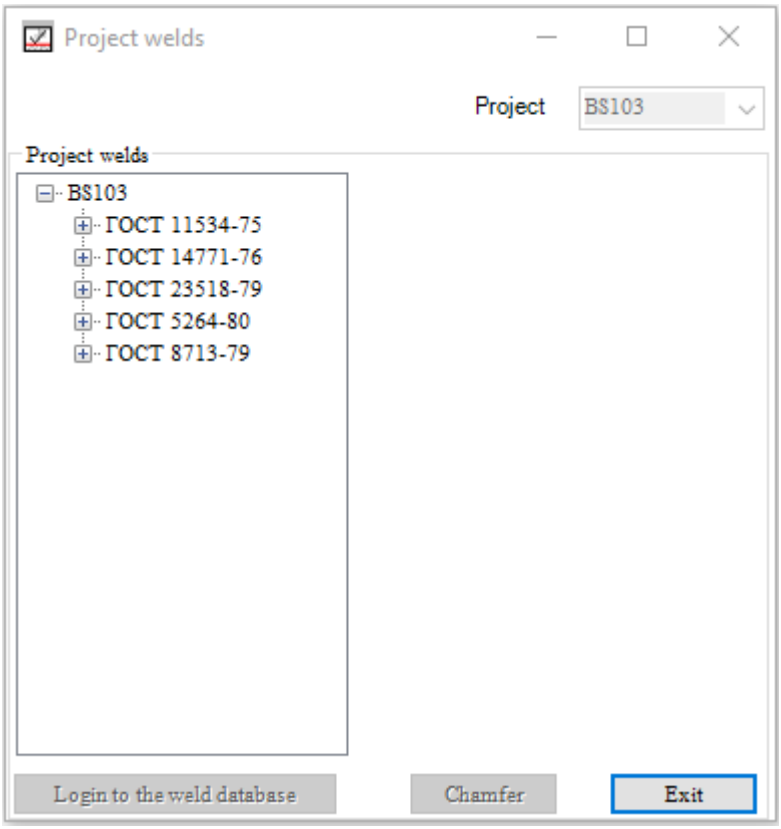


Figure 199. Project welds dialog box

The window is designed to form the Database of welds of the project based on the parameters of welds specified in ГОСТ and calculation of chamfer parameters on the edge of the part.

Exit the window using the Exit button.

The N-Ship+ system comes with a database of welds for the most used in shipbuilding ГОСТ for welds. There are five standards parameters: ГОСТ 8713-79, ГОСТ 5264-80, ГОСТ 14771-76, ГОСТ 11534-75 and ГОСТ 23518-79. Parameters of the welds specified in the standards are copied to the database of the current project from the catalog of the N-Ship+ system at the first call of the window “Project welds” or the command DETAILS MODEL... .

In the upper right corner of the “Project welds” window there is a list of database projects in which you can activate the project.

Attention: The list contains only projects of the active user, which have a subdirectory /Svarka/ in the directory <disk name>/NSHIP/Projects/<project name>.

When pointing to the ГОСТ item in the weld tree, the Project welds window has the view shown in Figure 200. The message box on the right shows the full name of the document.

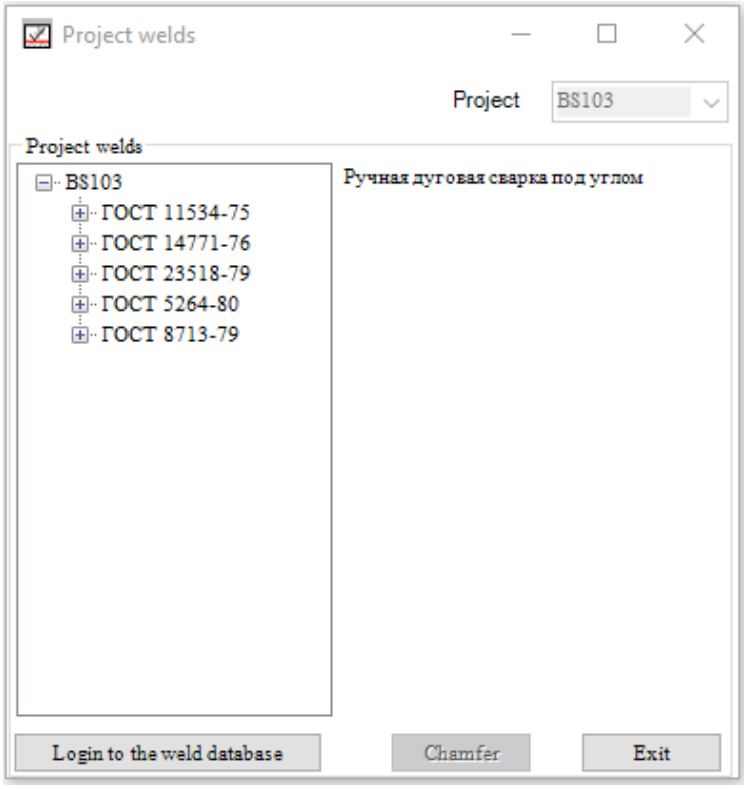


Figure 200. Project welds dialog box

After opening the ГОСТ item in the tree of welds and selecting the item with the weld code or the item with the weld view, the “Project welds” window has the view shown in Figure 201. The view window shows a primitive appearance of the weld.

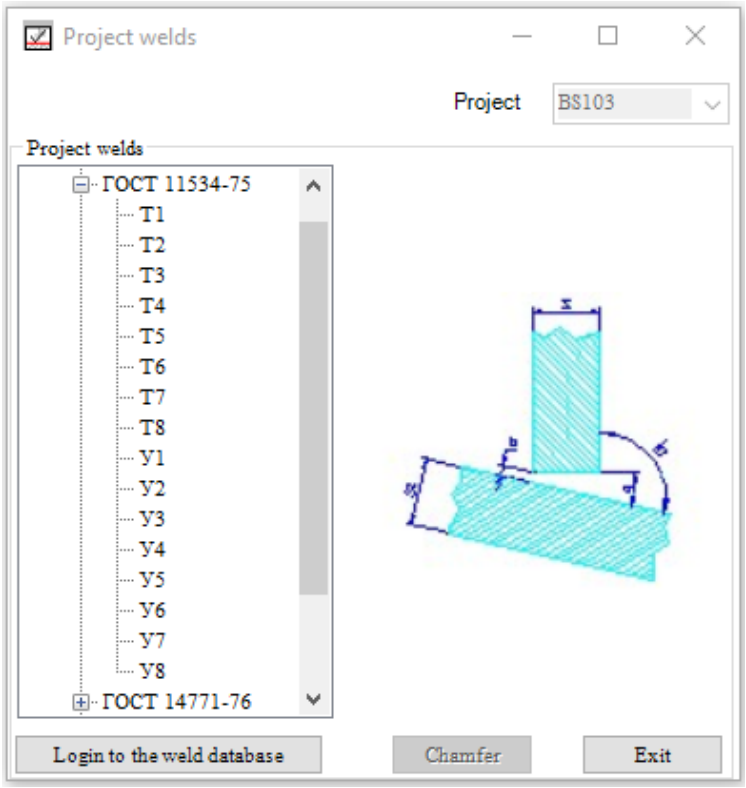


Figure 201. Project welds dialog box

In order to display the sketch of the weld edge cutting (copying if necessary or qualitative viewing), it is necessary to “double-click” the code or type of weld in the tree. Exit from the weld drawing by double-clicking on the code or weld type again.

The **Chamfer** button is activated only for welds with a weld cut. By clicking on the **Chamfer** button the procedure of forming a chamfer block on the part field in the active NanoCAD window is started.

Attention: In the active window there must be a pattern contour on the KBAS layer.

In order to add a new document to the weld database, it is necessary to specify the item with the project name in the weld tree and click on the **Login to weld database** button, a new window shown in Figure 202 will be called.

In this window are available for input field **Document name** and below the input field for a brief description of the document, for example, the number of the drawing, for which copying from ГОСТ can create a selection of welds of the drawing. It is necessary to enter data in the input fields, after entering the brief description of the document it is necessary to press the enter key, the **Write it down** button will be activated.

By pressing the **Write it down** button, this document is written to the welds database of the project.

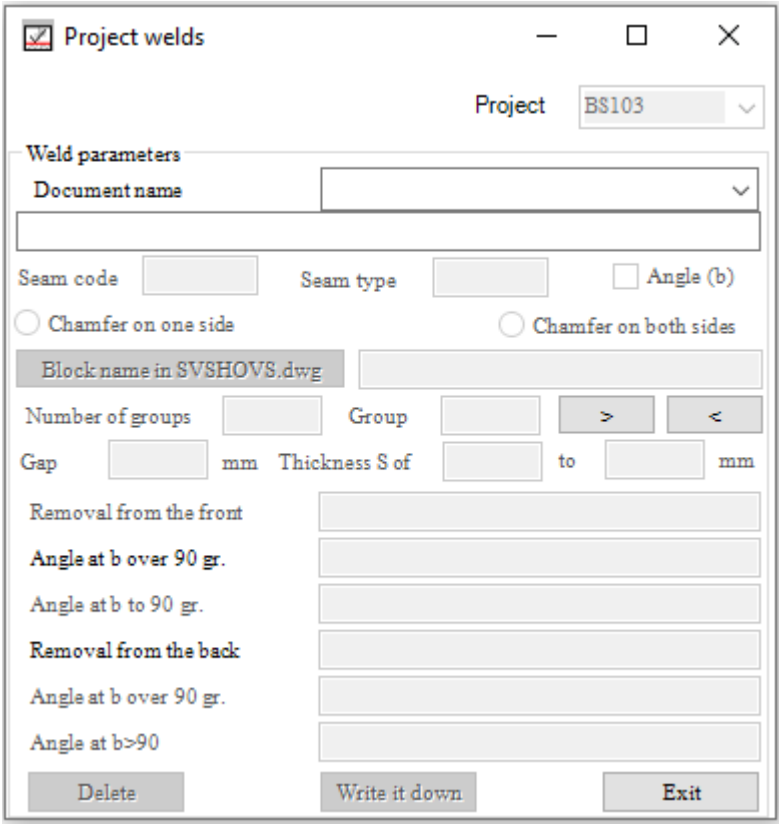


Figure 202. Project welds dialog box

When selecting an item with a weld code or weld type in the weld tree and clicking the **Login to weld database** button, the window shown in Figure 203 will be called up.

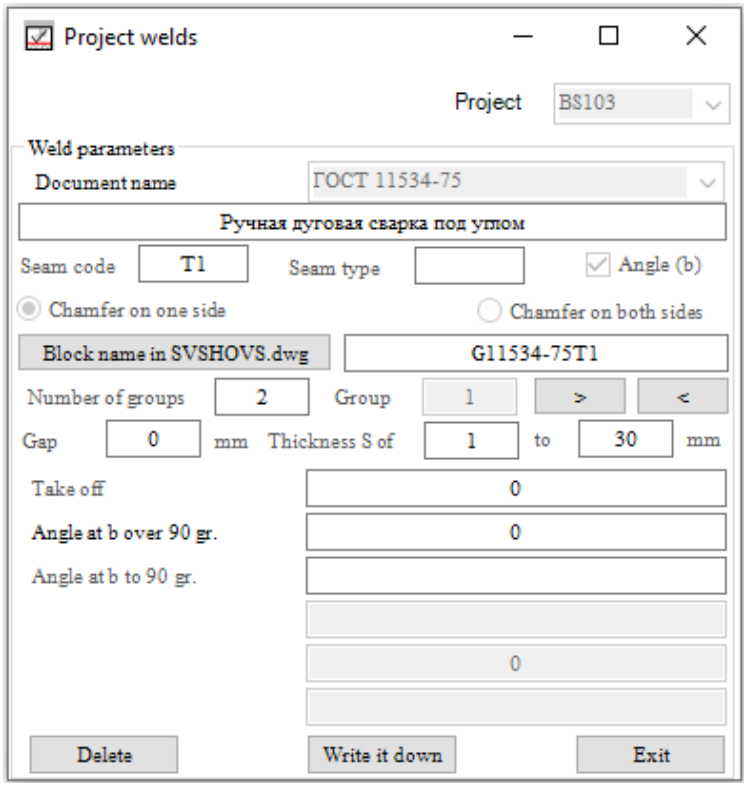


Figure 203. Project welds dialog box

This window shows the weld parameters and unlocks the **Block name in SVSHOVS.dwg**, **Delete** and **Write it down** buttons.

In the available data entry buttons you must enter your own data for new welds or for correcting the data of an existing weld, if necessary.

When you click on the **Block name in SVSHOVS.dwg** button, a window for selecting a .ipg file from the <disk name>/NSHIP/Projects/<project name>/svarka/shovjpg/ directory is opened.

The formulas for calculating the chamfer parameters **Take off** and **Angle** depend, as a rule, on the thickness (s), which in turn varies within specified limits, which is specified by the number of thickness change groups. Formulas for calculating the removal are written according to the rules of the Lisp language.

If you press the **Write it down** button, the data will be written to the weld database of the project.

If you click the **Delete** button, the weld data will be deleted from the project weld database.

Clicking the **Exit** button will exit the window and return to the previous window with the project welds tree.

The procedure of copying a weld from ГОСТ is as follows:

- create a new document;
- specify in the tree the weld to copy
- move (drag and drop) the weld with the right mouse button pressed to the new document;
- specify in the pop-up window in the input field the number of the weld in the new document.

9. WORK WITH THE DATABASE OF STRUCTURAL UNITS OF THE PROJECT

9.1. **NODES TABLE...** command

The command **NODES TABLE...** of the **MDET** module is intended for forming information on structural units in the project database.

As a rule, the designer develops an album of structural units in the form of a dwg file, as shown in Figure 204.

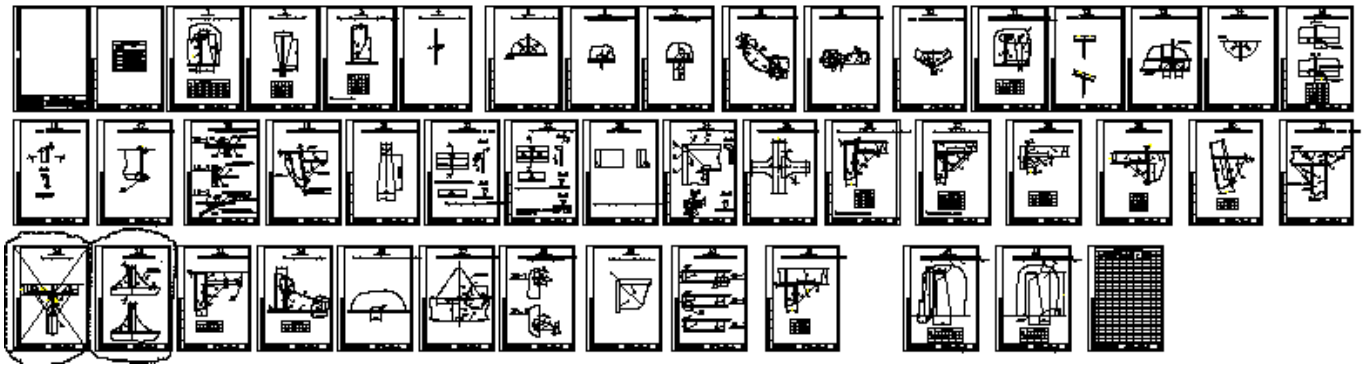


Figure 204. Album of structural units

Working with such a file is not very convenient. It is suggested to make each page as a block and form a tree of blocks, which would be convenient to work with.

When calling the command **NODES TABLE...,** a procedure is called, which first checks the presence of the node table file in the project database, and if there are nodes, the window shown in Figure 205 is opened, otherwise the window shown in Figure 206 is opened.

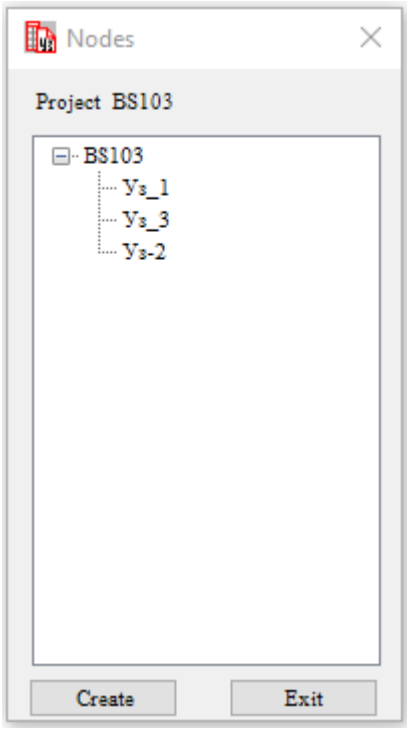


Figure 205. Nodes dialog box

Selecting a node brings the corresponding block to the screen, and selecting a node with the project name in the tree or double-clicking the node brings the source file to the screen.

Attention: Selecting a node with the right mouse button in the window shown in Figure 205 starts the procedure for deleting the node.

Clicking the Create button brings up the window shown in Figure 206.

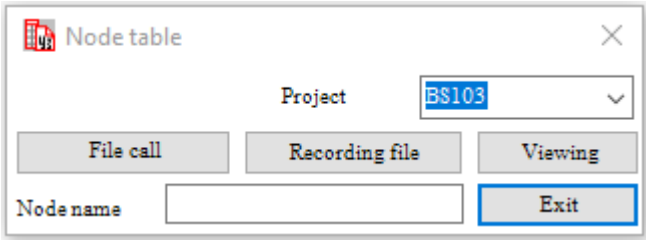


Figure 206. Nodes dialog box

The Project drop-down list contains projects registered in the system database, the active project is shown in the list. Sometimes it is necessary to view the node table of another project in order to copy a node; this can be done by specifying another project in the **Project** list, if it contains a node table.

When you click the **File call** button, the window shown in Figure 207 is displayed.

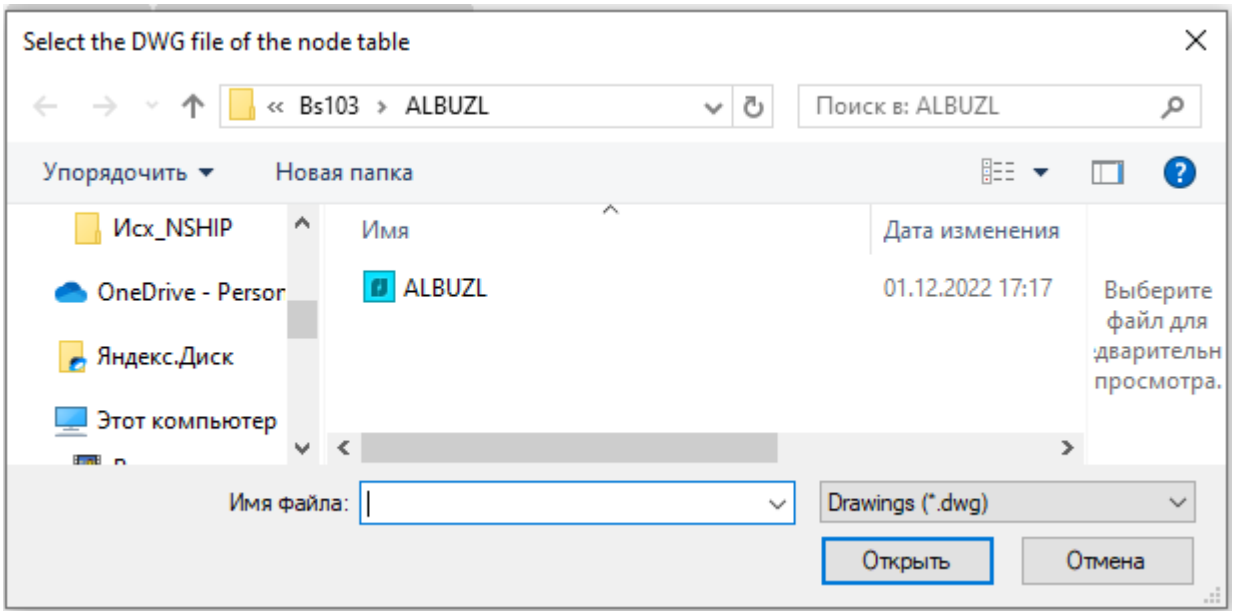


Figure 207. Окно вызова dwg-файла

In this window it is necessary to specify the source file created by the designer and click the **Open** button.

If there are no nodes in the file, then to create a block of each node, it is necessary to enter the name of the node in the **Node name** input field and press **Enter**. The procedure of forming a block with the specified name is started, at the same time a check for the presence of a node with this name in the drawing is performed and a request for selection of drawing primitives to be included in the block is issued. After block formation, the specified primitives are removed from the drawing (for convenience).

Attention: If you answer **Yes** to the request to replace a node, the node is deleted and the primitives included in the node are restored in the original file.

After all nodes are formed it is necessary to delete unnecessary primitives in the source file and click the Write file button. The procedure of writing the generated file to the project database starts, and the window shown in Figure 208 is displayed.

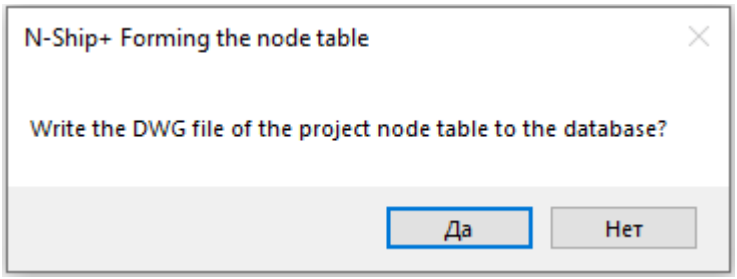


Figure 208. Request window

If you press the **Yes** button, the dwg file named ALBUZL will be written to the project database (directory .../Projects/project name/ALBUZL/...).

Exit from any window using the **Exit** button.

10. SERVICE FOR WORK IN THE MODEL

10.1. Window interface of the SERVICE command

Figure 209 shows the main submenu of the **SERVICE** command of the M-Detail module menu.

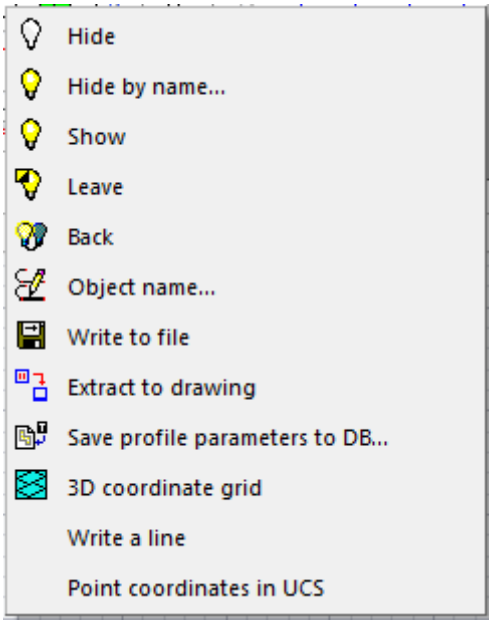
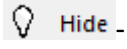
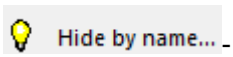


Figure 209. SERVICE command submenu



- clicking this icon starts the procedure of hiding the specified model objects.



- when you click this icon, the window shown in Figure 210 pops up.

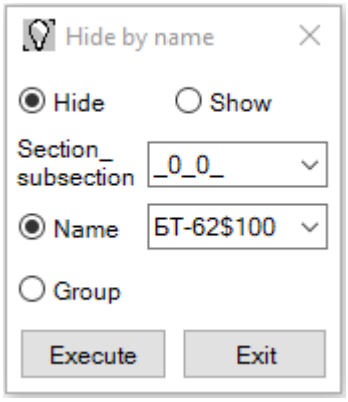


Figure 210. Hide by name dialog box

This window contains:

- - two switches - **Hide** and **Show**;
- - list – **Section_subsection**;
- - two switches - **Name** and **Group**;
- - two buttons - **Execute** and **Exit**.

The lists reflect the layers of the current dwg file, which are formed according to the rules of model lines naming in the N-Ship+ system.

If the **Name** pointer is active, you can select **Section-Subsection** and layer name from the corresponding lists. If you click the **Execute** button, the primitives of the active file will be hiding if the **Hide** switch is active and showing if the **Show** switch is active.

When the **Group** switch is active, the window looks as shown in Figure 211.

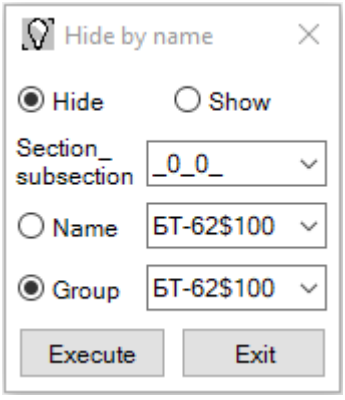


Figure 211. Hide by name dialog box

In this window you must specify the **Section_Subsection** and two names that will set the boundaries of the name group from the upper and lower list. The upper list shows the initial name and the lower list shows the final name of the primitives in the specified **Section_Subsection**. The lists are sorted in ascending order of names.

If you press the **Execute** button, the primitives of the active file in the specified name range will be hiding or showing.

Exit the window using the **Exit** key.

Show - pressing this icon causes the procedure of showing hiding objects by **Hide** icon.

Leave - clicking on this icon causes the procedure of hiding on the screen of all NanoCAD primitives except those specified by request in the command line.

Back - clicking on this icon causes the procedure of showing on the screen of all NanoCAD primitives except those specified by request in the command line.

Object name... - clicking on this icon brings up the screen shown in Figure 212.

When the From Name Table switch is activated, the window contains two lists:

- - a main list of suggested standard names;
- - an additional list, which can be supplemented with your own standard names, or unused names can be deleted from it.

For this purpose, the “Additional list” window has two buttons **Add** and **Delete**.

If you press the **Add** button, the name previously specified in the **Type** field will be added to the additional list. If you press the **Delete** button, the highlighted name is deleted from the additional list.

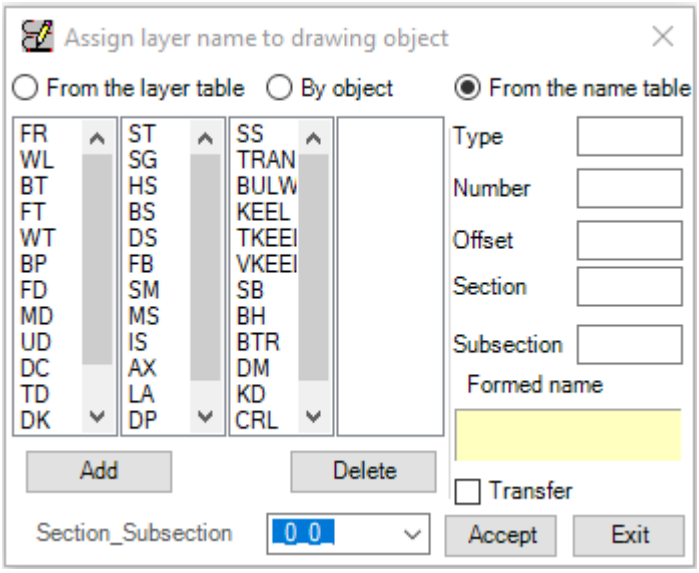


Figure 212. Assign layer name to drawing object dialog box

When you select a name in any list, it is copied to the **Type** field.

Under the type (name) there are fields **Number**, **Offset**, **Section** and **Subsection**. In these fields you should enter values and press **Enter**, the field - **Formed name** will be filled.

Attention: The model line name has the following parts:

- type (SHP, VL, BT, VP, PB, PER, etc., only Russian and Latin letters are allowed, which are automatically converted to upper case);
- main line number (integer, may have a minus sign);
- offset from the main line (non-negative integer. If it is zero, the offset is not reflected in the name);

- section (integer in the range from -32768 to 32767, plus sign is not allowed);
- subsection (integer in the range from -32768 to 32767, plus sign is not allowed).

The **Section** and **Subsection** fields can be filled by selecting the corresponding value in the **Section_Subsection** list, which is formed by existing objects in the model.

Attention: List **Section_Subsection** can be replenished by entering values in the **Section** and **Subsection** fields and pressing enter.

Activation of the **Transfer** pointer means that simultaneously with assigning a name to an orthogonal section, it will be transferred to the level defined by the spacing table.

When the name formation is finished, you must click on the **Accept** button, after which the system will ask you:

->Indicate the line of the frame:

if these are orthogonal sections of the model, or

->Specify the object of the drawing:

otherwise.

The user must specify the named objects by any means allowed in the nanoCAD system environment. After specifying the objects, they are transferred to the layer with this name. If the layer with the required name did not exist before, it is automatically created.

When the **By object** switch is activated, the window looks as shown in Figure 213.

The switches **Frame**, **Waterline**, **Buttock** and **Object** are used to select the appropriate type of line from the drawing after clicking on the **Specify** button. If the specified line does not correspond to the switch type, the system displays a message about it, otherwise the **Type**, **Number** and **Offset** fields are filled in by the name of the layer on which the line is located. The fields **Section** and **Subsection** can be filled by selecting the corresponding values from the **Section_Subsection** list.

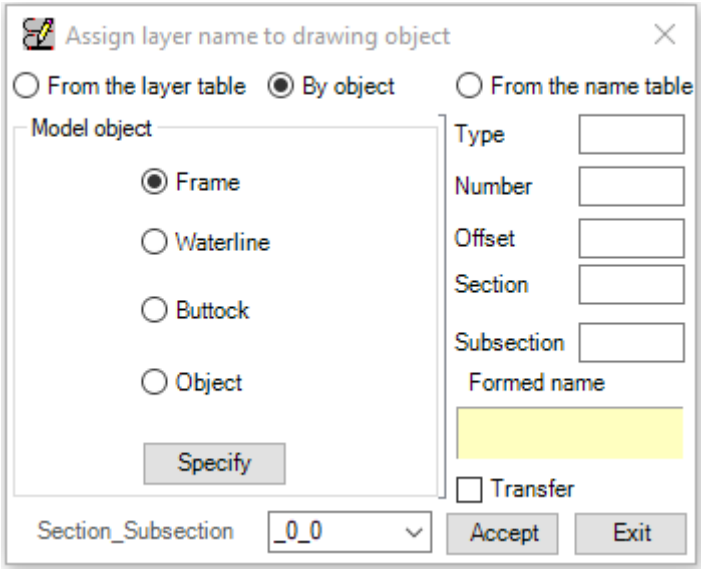


Figure 213. Assign layer name to drawing object dialog box

With the **From the layer table** switch activated, the window looks as shown in Figure 214.

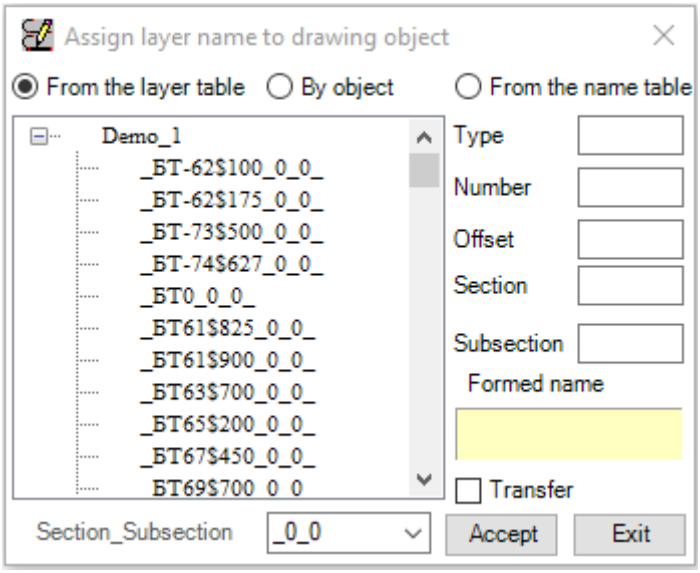
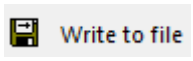


Figure 214. Assign layer name to drawing object dialog box

This window shows a tree of layers, which are formed according to the rule of forming line names of the N-Ship+ system model. All layers are sorted by section and subsection, the list of which is shown below the layer tree window. If you specify an item in the layer tree, the layer parameters are copied to the input fields **Type**, **Number**, **Offset**, **Section** and **Subsection**.

Exit the 'Assign layer name to drawing object' window by clicking the **Exit** button.



- clicking on this icon brings up the screen shown in Figure 215., in which you must select the directory and assign the name of the .dwg file in which the drawing objects will be saved.

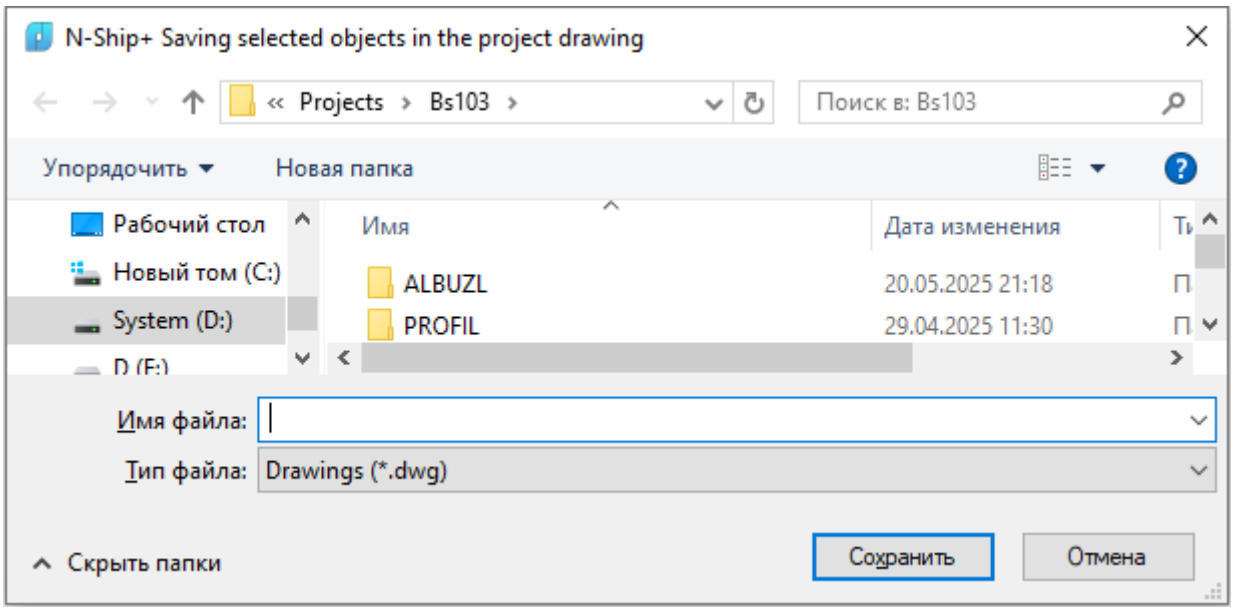
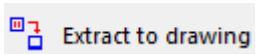
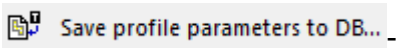


Figure 215. Window for saving a dwg file



- clicking on the icon triggers the procedure of selecting objects in the drawing. After that a new window opens and the selected objects are copied into it.



- clicking on the icon calls for execution of the command intended for calculation and recording of the profile cross section parameters in the materials database. The window shown in Figure 216 pops up on the new drawing field.

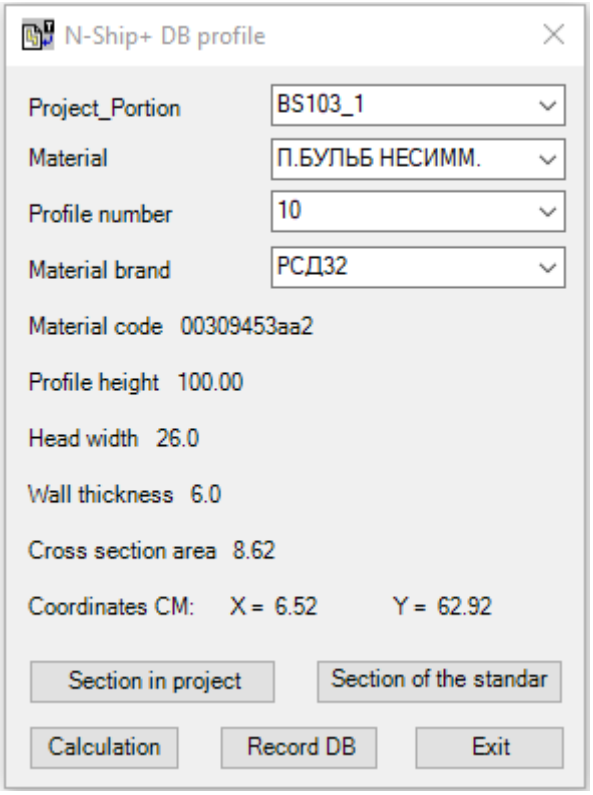


Figure 216. Database profile dialogue window

In the control window the necessary profile number is selected, parameters are calculated and displayed in the window, section parameters are recorded in the materials database.

If the profile active in the window does not have a cross-section line in the database, the window shown in Figure 217 pops up.

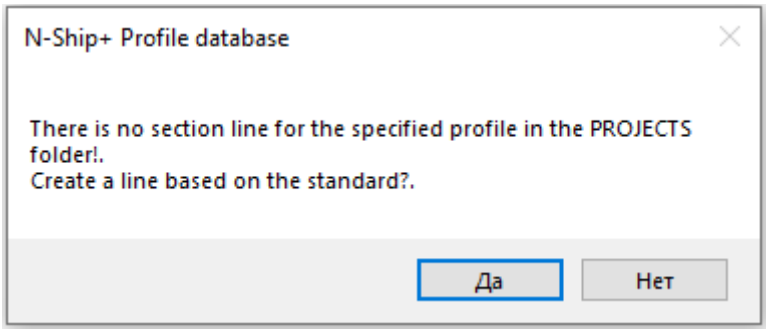


Figure 217. Request window

If the **Yes** button is pressed, the procedure of reading the cross-section line from the database is started and if there is such a line in the database, it is drawn on the screen. If there is no such line, it is necessary to draw it under the following conditions:

- when specifying the profile orientation in the window interface, orient the profile as it is specified in the corresponding assortment (e.g. for an asymmetrical hollow section - thickness to the right of the 0Y axis, head turned to the right);
- it is recommended to set the profile at point 0,0;
- the position of the wall should be accurately directed along the 0Y axis.

Pressing the button **Calculation** starts the procedure of formation of cross section parameters of the profile by line.

The dialogue of the graphical editor at pressing the button **Calculation**

-> Specify the contour line of the profile section

It is necessary to specify the contour line of the profile section.

-> Specify a point on the wall to set the thickness:

It is necessary to specify an arbitrary point on the profile wall.

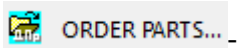
-> Specify a second point on the opposite side of the wall:

It is necessary to specify the second point on the opposite wall of the profile.

The necessary calculation will be performed and the calculated parameters will be displayed in the window.

Recording to the materials database is performed by the button **Record DB**. The recording procedure asks in the window whether it is necessary to save the section contour line along with the section parameters. If the user confirms the necessity of saving, the file of the current drawing with the contour line is saved in the folder ...NSHIP\Projects\<project name>\profil\. The name of the file is the parameter profile material code from the materials database. The profil folder is created automatically.

Attention: the **Exit** button closes the current open drawing.



- clicking on the icon brings up the screen shown in Figure 218.

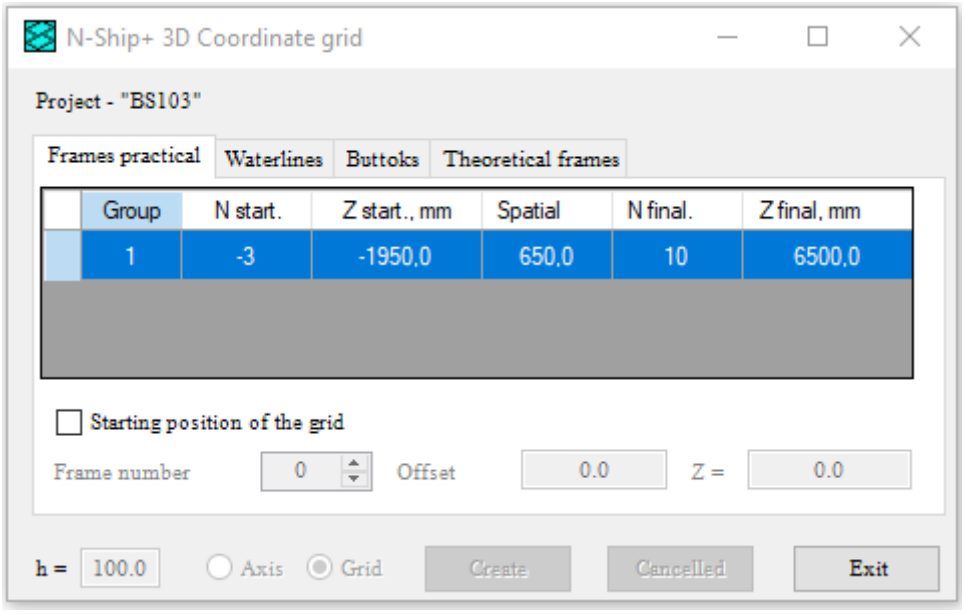


Figure 218. Coordinate grid dialogue box

This window contains 4 tabs, each of which contains a window reflecting the corresponding spacing table and a set of fields for specifying the coordinate grid position in the model. To activate the fields, click on the Initial grid position pointer.

When you specify a section number, the section coordinate is defined, which can be corrected by specifying an offset. The coordinate can be set directly in the input field.

Attention: The coordinate is checked to make sure that the value is within the spacing table.

After specifying the coordinate, the **Create** button is unlocked, when you click on it, the coordinate grid block is set in the model.

The type of the coordinate grid is defined by the **Axis** and **Grid** switches.

If you specify **Axis**, then the input field h is unlocked for specifying the value of strokes of the corresponding sections on the axes.

If you press the **Cancelled** button, all the meshes created in the model will be deleted.

Click the **Exit** button to close the 'N-Ship+ 3D Coordinate Grid' window.

The **Write a line** command is intended to form an inscription on the MARK layer with the name of the layer on which the line is located.

The command dialogue :

-> Specify the lines for the lettering :

The **Point coordinates in UCS** command is used to form inscriptions with the coordinates of points in the specified coordinate system on the DIM layer.

Command dialogue:

->Specify the length of the line :

->Number of decimal places :

->Specify the origin of the new coordinate system :

->Specify the rotation angle of the new coordinate system :

->Specify a point :

->Specify point :

Лист регистрации изменений									
Изм.	Номера листов (страниц)				Всего листов (стр.) в документе	Номер документа	Входящий № сопроводит. докум–мента и дата	Подпись	Дата
	измененных	замененных	новых	аннулированных					