CAD/CAM System N-Ship+

Version 5.0

Model Structure

Creation of internal structures and superstructures

User manual

NSHIP.00009.005-2025

Sheets 50

Saint Petersburg

2025

ANNOTATION

The document is a reference manual for work with the module **Structure** of the **N-Ship+** system. The manual includes description of menu, commands, user interface, themes of interaction with other modules of the system.

Document is designed for specialists who run **N-Ship+** system for construction and technological preparation of ship hull production and have practical experience of using nanoCAD or AutoCAD graphical system.

N-Ship+ is informationally compatible with the systems **Ritm-Ship** (AutoCAD), **R-Ship+** (AutoCAD), **B-Ship+** (BricsCAD).

Contact data:

Mobile: **+7 921 7561226** (Nikolai Poleshchuk)

Email: **npol50@yandex.ru**

Web: **http://poleshchuk.spb.ru/cad/2016/nshipe.htm**

CONTENTS

1. GENERAL INFORMATION

The described program (hereinafter module **Structure**) allows building in 3D model with prepared hull forms lines for ship structures on design stage and on technological preparation stage for defining dimensions and forms of the parts. Module efficiency results in quick recalculations of structures with similar but different data. Each structure has its own data file and can have various versions and revisions. Deck rebuilding can be done in the same session. Then recalculation of bulkhead attachted to this deck will result with a new geometry.

For shortness in the document system N-Ship+ will be named N-Ship.

2. APPLICATION CONDITIONS

Module works in the environment of graphical editor Platform nanoCAD, requires prepared and initialized ship wireframe model DWG file prepared and initialized with module **Model** of N-Ship. Hull forms lines (frames, waterlines, buttocks) must be created in the corresponding ship UCSs. Hull lines and lines of other structures (platforms, decks, etc.) taking part in building process must be 2D polylines on corresponding layers. Layer names must satisfy to N-Ship naming rule structure:

<_> + <*type*> + <*number*> + <*addition*> + <_> + <*partition*> + <_> + <*subpartition*> + <_>*.*

Here:

<_> — subscription symbol (“_”);

<*type*> — type name, only latin letters (FR, WL, BT, UD, BH etc.);

<*number*> — number (integer -32768 to 32767) of the nearest main line;

<*addition*> — empty string for zero addition (e.g. it is name of main frame) or string like *“$”* + <*positive number*> (e.g. if it is name for intermediate frame moved forward);

<*partition*> and <*subpartition*> — integer numbers from -32768 to 32767.

For example, layer name for 25th main frame: _FR25_0_0_. For additional (intermediate) frame moved 100 mm forward from main frame 25: _FR25$100_0_0_.

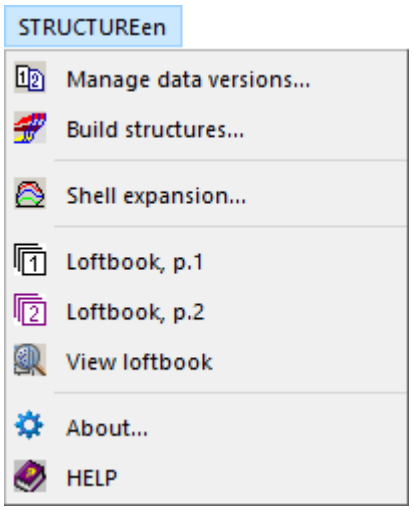
In other localizations types are written in corresponding language, e.g. Russian: ШП, ВЛ, БТ, etc.

3. FUNCTIONAL DESIGNATION

Module is intended for building decks, platforms, superstructures, longitudinal bulkheads, seam lines and longitudinal girder attachment lines (hereinafter structures) based on source data for structures and ship hull forms 3D model.

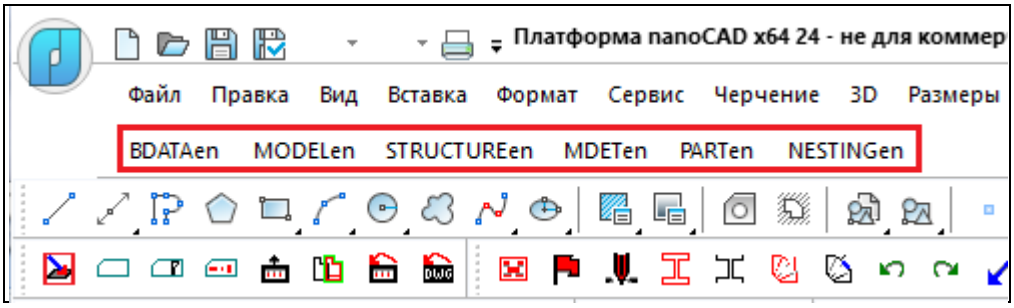
4. USER INTERFACE DESCRIPTION

Module **Structure** needs drop-down menu **STRUCTURE** at the end of which a suffix defining interface language is added: **STRUCTUREen** (English), **STRUCTUREru** (Russian), etc. (dr. 1). For shortness in the document menu will be named without suffix (**STRUCTURE**).



Drawing 1. Menu **STRUCTURE**

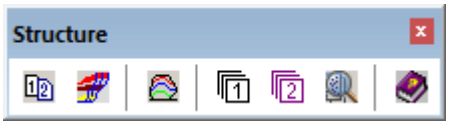
Loading menu is described in administrator manual (N-Admim_en.pdf). Dr.2 shows menu line with loaded N-Ship English pull-down menus.



Drawing 2. Loaded menus of N-Ship

Полный набор функций модуля **Structure** доступен также с помощью панели инструментов **Structure** (рисунок 3).

Full set of module **Structure** commands is also accessible from the toolbar **Structure** (dr. 3).



Drawing 3. Toolbar **Structure**

Russian language interface language of module **Structure** is used when in module **Bdata** there was applied menu command **BDATA > UI language**.

5. USED HARDWARE AND SOFTWARE

Programs of module **Structure** are oriented to personal computers. Recommended configuration of hardware and systems is as follows:

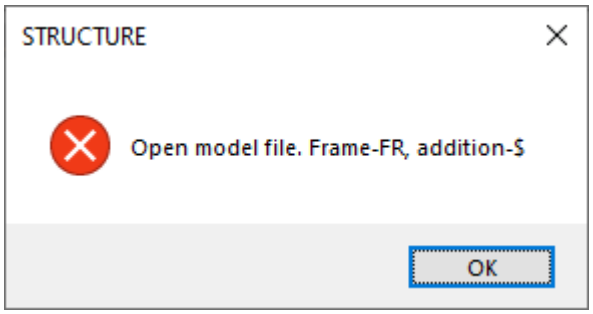
- Intel Core Duo 2.2 GHz, RAM 16 Gb, HDD 1 Tb, Video Card 128 Mb, monitor 21'';
- operating system Windows 8.1, Windows 10, Windows 11 (64-bit);
- Platform nanoCAD of version 24 or greater.

6. LOAD AND START

Normal functioning of module **Structure** in the nanoCAD environment supposes preliminary setting special Windows registry parameters, saving path to the N-Ship+ root folder (by default *D:\NSHIP* is suggested) and path to projects folder *NSHIP\Projects*. Initial setting is done in the process of installing system and it can be modified under specific work conditions.

For launching main calculation procedure it is necessary to select menu item **STRUCTURE > Buid structures**.

On start program verifies if current model DWG file has any non-empty frame line layer, i.e. layer with prefix _FR, with digit or minus next. If no such layers are found then a message is issued, suggesting opening model file (dr. 4).



Drawing 4. Message on invalid model

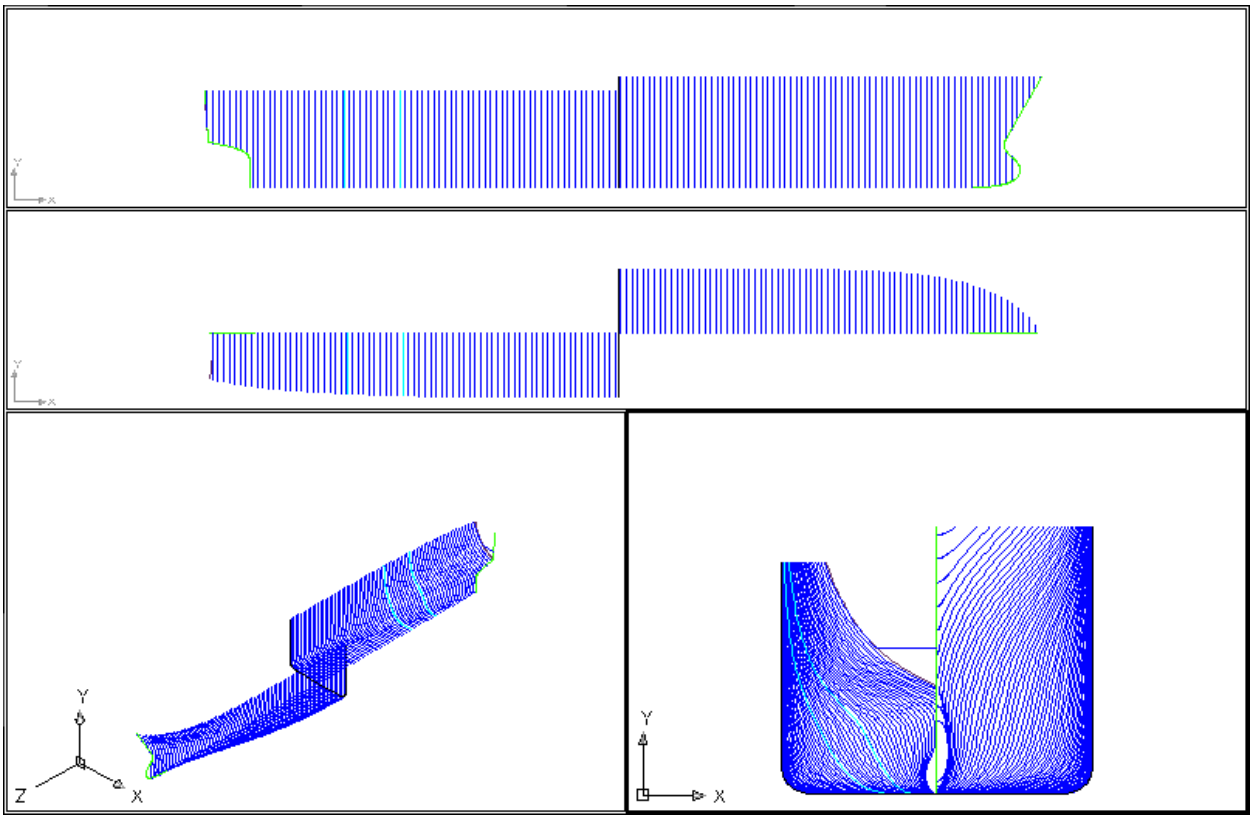
Click **OK** to stop work. User must open model file with prepared hull forms and start running once more. On dr. 5 there is a sample 3D wireframe model.

7. INPUT DATA

Input data are divided into some groups, with individual dialog boxes.

7.1. General data

Module **Structure** software allows working with different projects, and each project has its own folder located in the common folder for all the projects that was indicated in system settings. Inside common folder there a folder with project name (for example, *D:\NSHIP\Projects\71144* for project 71144*)*. Data files and log files of module **Structure** are located in subfolder *Steel* of the project folder (e.g. *D:\NSHIP\Projects\71144\Steel*).



Drawing 5. Sample file with prepared hull forms

Inside one project user has possibility to create up to 99 versions of data for each ship structure, enabling multiple kinds for one structure surface. Names file must apply the rule: *steel_<nn>.dat*, where *<nn>* is a two-digits version number, e.g.: *steel_08.dat*. Version 00 (*steel_00.dat*) is called *basic* and cannot be deleted.

Therefore there is a procedure to set module **Structure** for work with required version. In case of a new project user must run some steps for setting files and parameters. Creating new project in done with menu item **STRUCTURE > Manage data versions**. There are generated spacing table files (with extension *.&sp*, *.&st*, *.&bp*, *&wp*) and general parameters file (*project.par*) that are initialized by standard (default) data. That's why contents of these files must edited with real project data.

For editing general parameters file and spacing tables use menu items **MODEL > General parameters** and **MODEL > Spacing tables** (see user guide for module **Model**).

Here is a sample contents of general parameters file (*project.par*) for project 71144:

General parameters file for project 71144

Bulk carrier

Design bureau

Classified

17.06.20

17.06.20

LMAX=127.00

LPP=137.00

B=6.35

BMAX=11.43

H=19.50

T=5.40

TM=5.20

NAME=Genhull

[Specific]

XDirection=1

*SternName=_BT0_0_1_*

*StemName=_BT0_0_2_*

*TransomName=_TRANSOM_0_0_*

For normal functioning of module **Structure** the following parameters must be present:

LPP — length between perpendiculars, m (in the sample — 137.00);

SternName — sternframe (aft profile) layer name (e.g. _BT0_0_1_);

StemName — stemframe (fore profile) layer name (e.g. _BT0_0_2_).

7.2. Calculation modes

Command of structures building can be run with menu item **STRUCTURE > Build structures** or with button  of toolbar **Structure**.

After command start dialog box **STRUCTURE Module. Project <71144>, version <0>** is opened (numbers of current project and data version are conditional) (dr. 6).

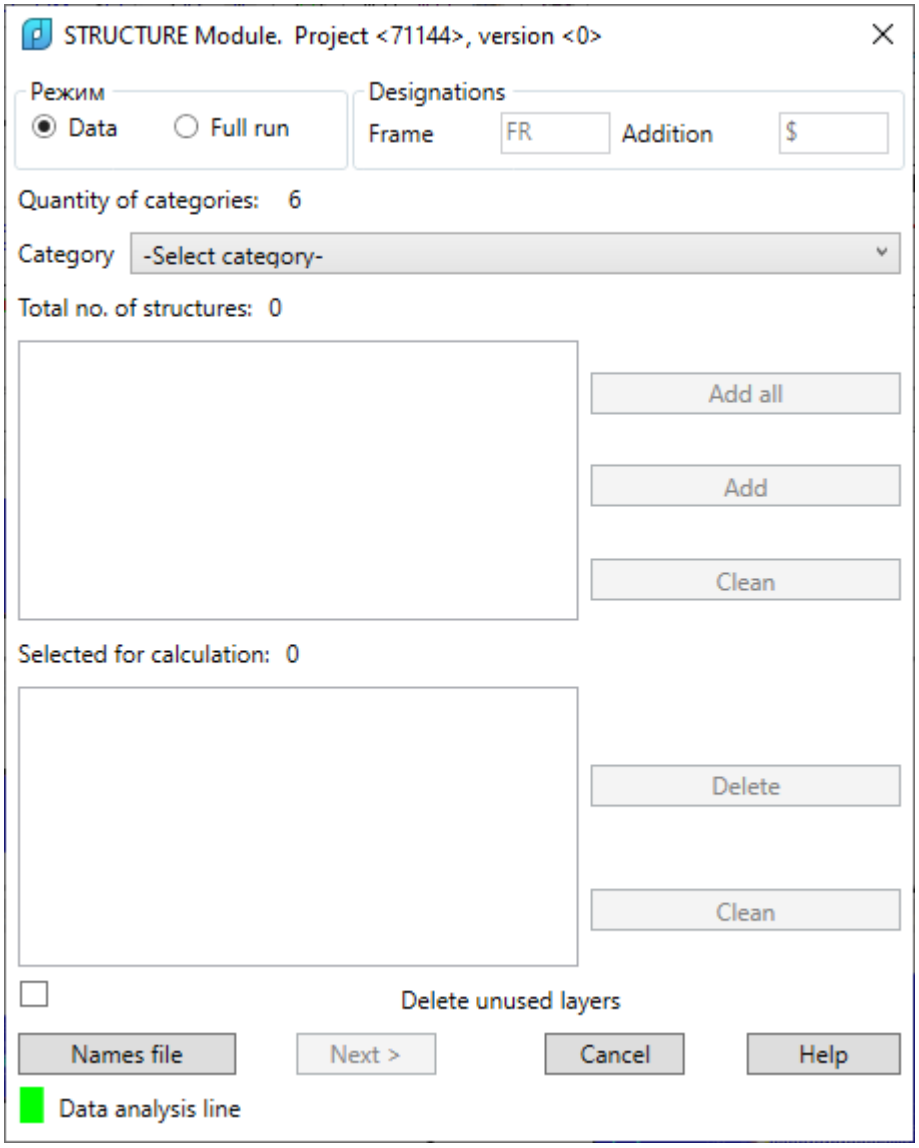
The window is intended for:

- setting mode of calculation (selective or full);
- selecting structures to be built or rebuilt in current session (in selective mode).

In the area **Mode** of window **STRUCTURE Module. Project <71144>, version <0>** there are two radio buttons:

- **Data** — work with data of selected structures (selective mode);
- **Full run** — building all the structures described earlier in names file *steel_<nn>.dat* (<nn> is a number of current data version).

- In the mode **Data** for structures selection there is drop-down list **Category** (with names of categories, into which all the ship structures are divided) and two lower listboxes. The listbox **Total no. of structures** displays all the structures of the selected category. The list **Selected for calculation** one displays categories that were selected for session. In this document the first listbox will be named *Category elements*, and the second listbox will be named *Selected elements*. Work with categories and selection of structures is possible only in the mode **Data**.

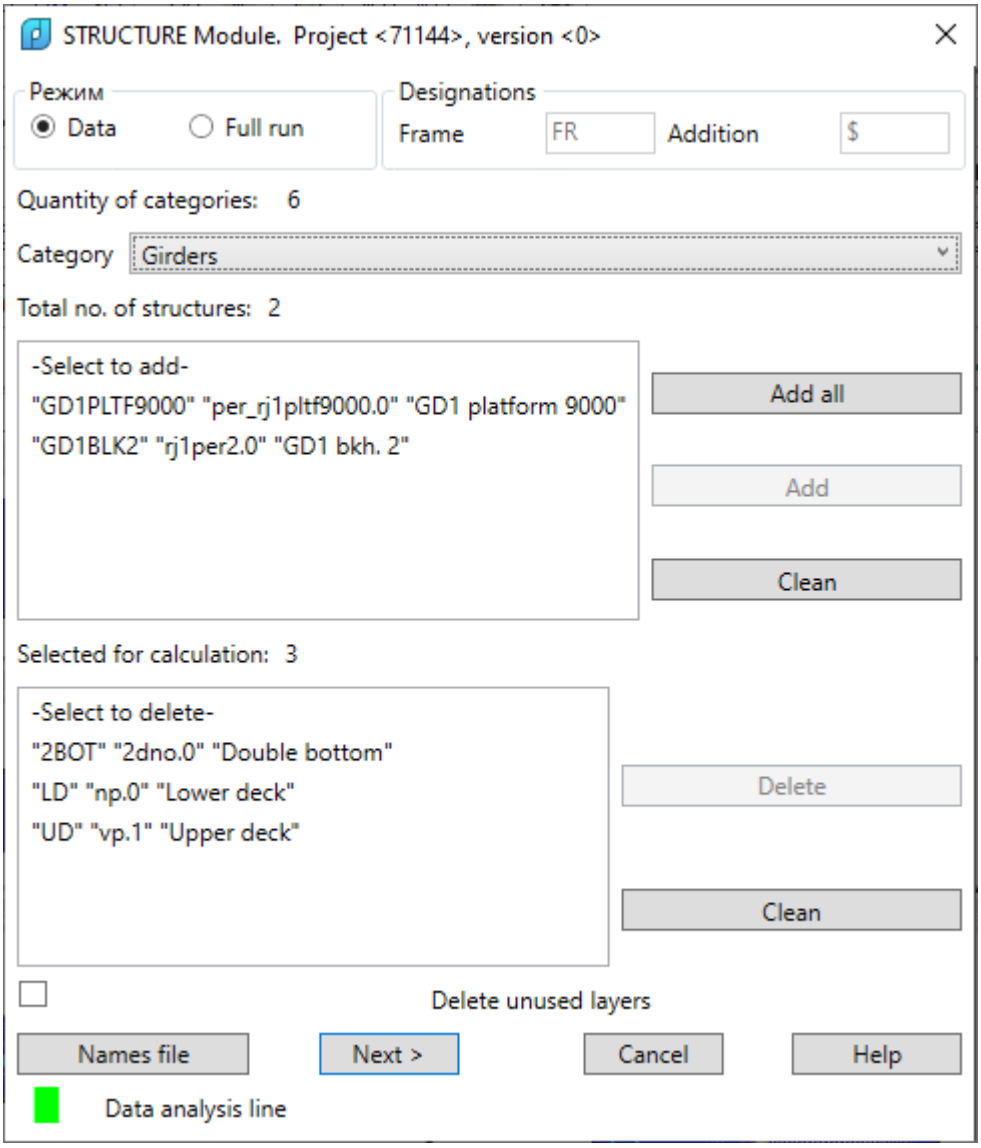


Drawing 6. Window of start for building structures

If to check box **Delete unused layers**, then empty layers will be deleted from the model.

Usually module is used for building structures only on one side, therefore it is sufficient to have hull form lines only for this side. Side can change on middle frame, it is convenient for proper view of the body plan to have aft frames on portside and fore frames on starboard side. For building structure lines on the opposite side then it is possible to mirror hull forms and to repeat building structures. To preserve lines of the previous side (e.g. for upper deck) clear checkbox **Delete structures** in the window **Upper deck. Service**.

In the mode **Data** (selective) in the window **STRUCTURE Module. Project <71144>, version <0>** drop-down list **Category** (dr. 7) shows text **-Select category-**. If click on the list will show categories already included into names file *steel_<nn>.dat*, e.g. *Decks*, *Platforms*, *Bulkheads*, *Girders*, *Holds*, *Seams side*. Over the list there is a text line with total quantity of records (categories).



Drawing 7. Selecting structures for calculation

If click on category name (e.g. *Girders*), then all members of this category will be copied to the listbox *Category elements*. In the line **Total no. of structures** quantity will be shown (on the drawing there are two structures: *GD1PLTF9000* and *GD1BLK2*). Next step is to select those structures that will be built in the current calculation.

To the right of this listbox there are three buttons:

- **Add all**, copies all the structures of the category to the lower listbox;

- **Add**, copies only selected structures to the lower list (for multiple selection click with pressed button **Ctrl** or **Shift**);

- **Clean**, clears the list.

To select one structure it is necessary to click on its name (marked structure will be shown with color) and then to click button **Add**. Name of selected structure will disappear from the upper list and will appear in the lower one. Line **Selected for calculation** will reflect total quantity of se-

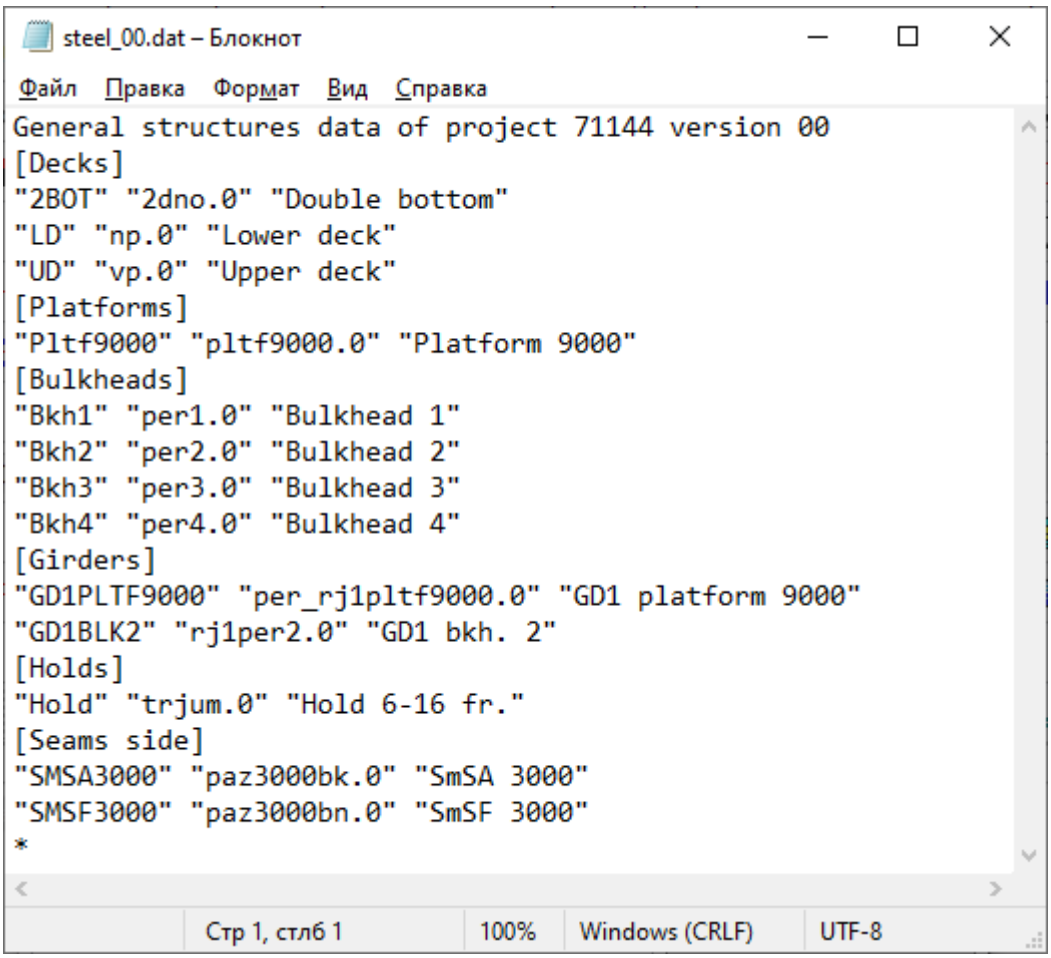
lected structures. If click button **Add all**, then all the elements of the list will be added and upper list will become empty.

Module monitors repetition of names and informs about it in the line **Data analysis line** with yellow square and message **Duplication**.

To the right side of the lower list there are buttons:

- **Delete**, deletes structures marked in list;
- **Clean**, clears the list of selected structures.

In the lower part there is a button **Names file**, click on it opens names file *steel_<nn>.dat* for viewing or editing with Notepad (or other default) editor of Windows (dr. 8).



Drawing 8. Structure names file

Each line corresponds to one structure or is a service line (comment, category title, etc.). Structure line consists of three names:

- short structure name used as a type in layer names, for example: “UD”;
- name of the file with current structure data (camber, sheer, etc.), for example: “vp.0” (extension equals to number of current revision of data for this structure);
- full structure name to be used in dialog boxes and tables, for example: “Upper deck”.

Button **Names file** must be used to get information about presence of structures for the mode **Full run**. To modify data a default editor (like Notepad or AkelPad) is called.

In window **Notepad** there is shown names file contents for the file version 0 (*steel_00.dat)*. In square brackets there are category names, accepted by user (in the sample **Decks**, **Platforms**, **Bulkheads**, **Holds**, **Seams side**). After category caption there go lines for structures of category (one line for one structure).

Categories lists ends with * (asterisk) in the first position. If list of categories contains only title then it must end with a line of two square brackets ([ ]) and a line with * (asterisk).

Structure names can be arbitrary except longitudinal bulkheads. File names for bulkheads must begin with "per" (for example *per1.1*, *per15.3*).

While describing structures in the names file user must consider that main structures must precede those structures that will be built on the basis of main ones. For example if longitudinal bulkhead is located between deck 1 and deck 2, then data of decks 1 and 2 cannot follow after the bulkhead data. Calculation is strictly executed in the sequence as they were selected in the dialog box on dr. 7 (for **Data** mode). If the correct sequence is violated then no intersection with decks will be found and in the log file there will be an info record. Quantity of categories and structure names is unlimited. In the full run mode all the structures that are written after line with * (asterisk) in the first position, will not be built.

If in the full run mode for a structure in the file *steel_<nn>.dat* there will be written name of missing file, then this file will be created without template (with default data), and a message will be added to log file *D:\NSHIP\projects\71144\steel\TAB\FF_saw_ko.$$$*, for example:

"Data in the file D: \NSHIP\projects\71144\steel\pltf7000.1 are entered by default"

Checkbox **Delete unused layers** gives opportunity to purge drawing from extra layers.

In the lower part of the dialog box there is **Data analysis line**, with a square icon, green at the beginning. This line is user for output error information and warnings.

Icon is colored to green when data are correct, to yellow for warnings, to red when serious error is found. In the last case cursor is returned into the field where the first error was discovered. To continue user must remove data error.

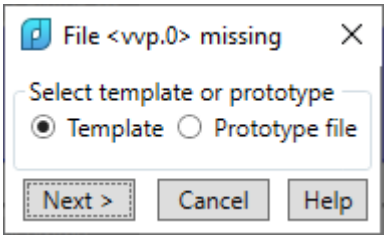
Up to the **Data analysis line** and after **Names file** button there are buttons more, two of them close window and **Help** calls necessary help topic.

Button **Next >** closes dialog and continues work. If in the **Data** mode no structures are selected then this button is disabled.

Button **Cancel** closes dialog and stops calculation.

7.3. Data template for structure

If in the names file *steel_<nn>.dat*> for some structure there was mentioned a data file that does not exist (e.g. *vvp.0*) then after clicking button **Next** in the main window the dialog box **File <vvp.0> missing** will be opened (dr. 9).

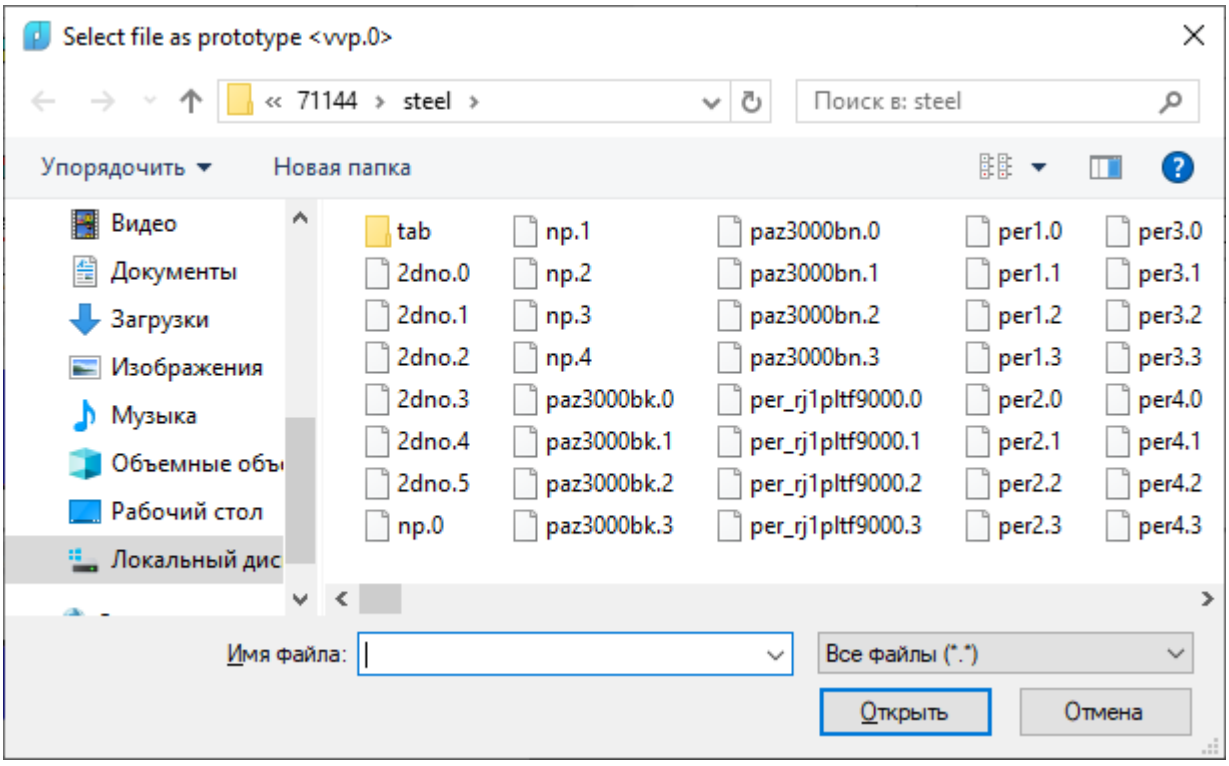


Drawing 9. Selection of template type

In the radio row **Select template or prototype** there are two radio buttons suggesting two kinds of replacement for structure data missing file:

- **Template**, forming file with standard default data;
- **Prototype file**, copying data from the prototype file to be requested.

If radio button **Prototype file** is selected and button **Next** is pressed then dialog box **Select file as prototype <vvp.0>** (dr. 10) and user must select the file (e.g. *vp.3*) from which data will be copied for created file vvp.0 (to be edited later).

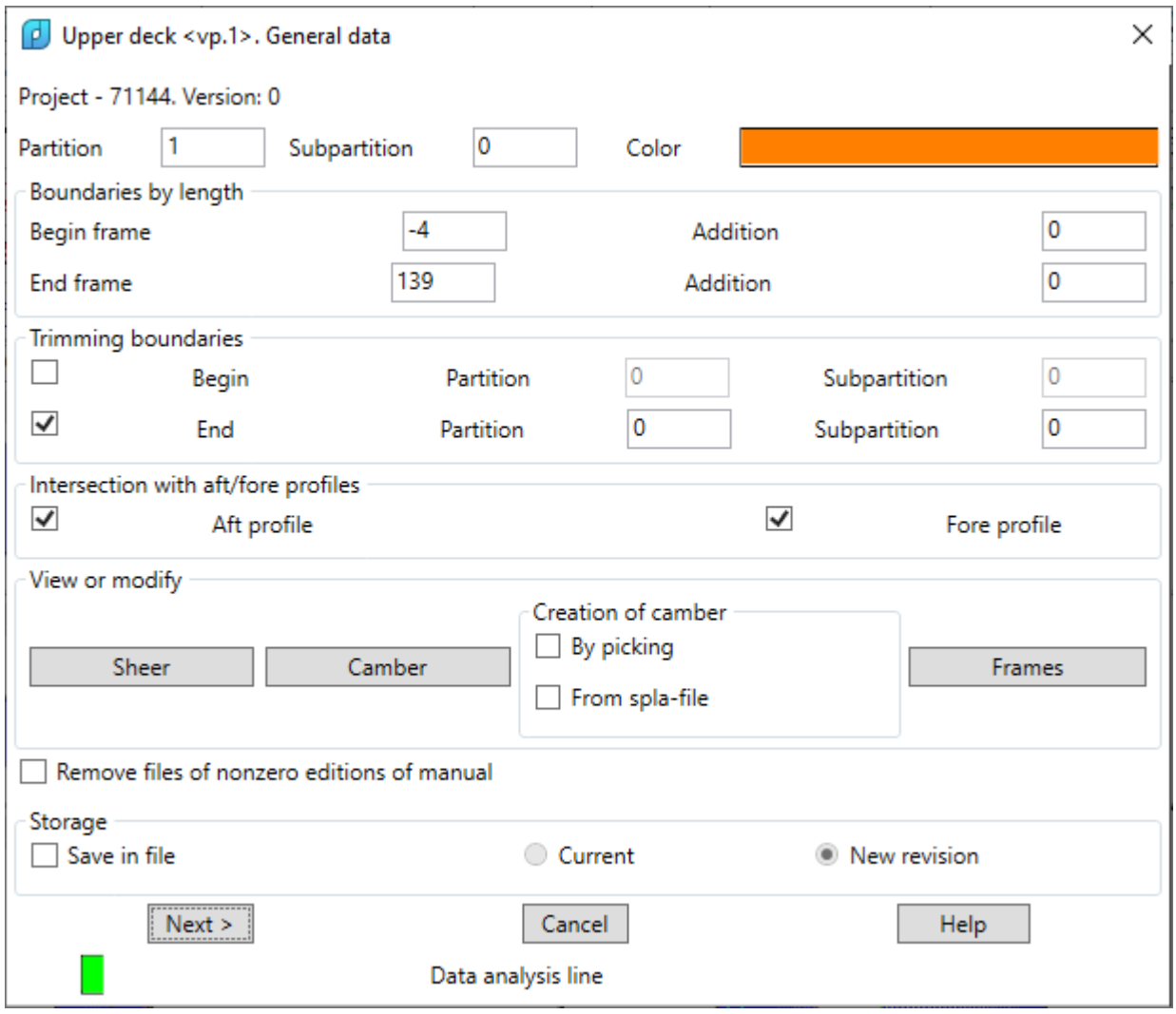


Drawing 10. Selection of file as prototype

Next steps in the **Data** mode open several dialog boxes. They will be discussed on sample upper deck.

7.4. General data for structure

Next is dialog box **Upper deck <vp.1>. General data** (dr. 11), if upper deck is selected in the list of structures to be built (let data be in vp.1).



Drawing 11. Window **Upper deck. General data**

In the upper part of the dialog there are three controls:

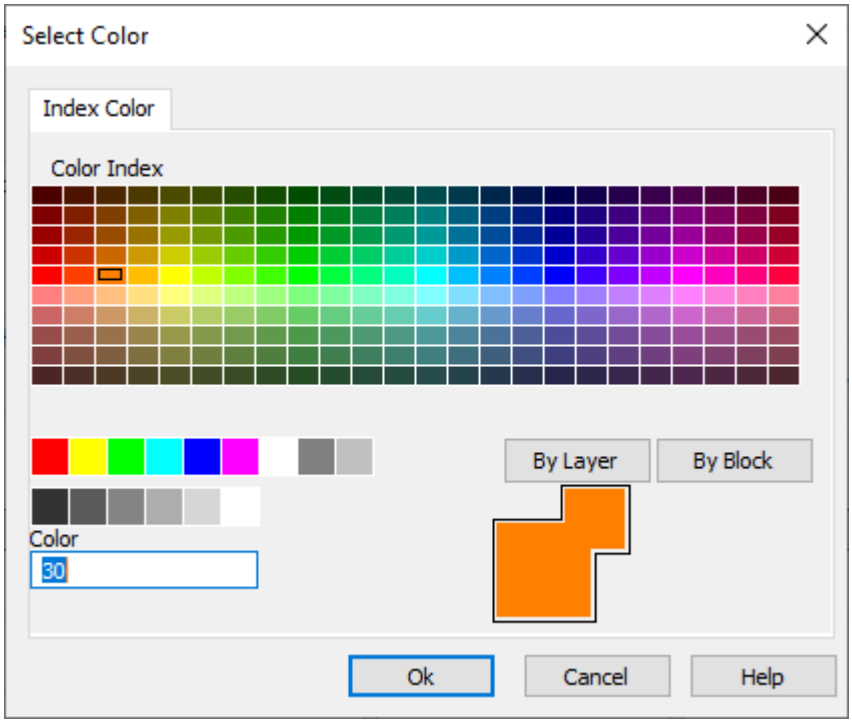
- **Partition**, partition for the side line of the structure, (e..g. _UP_**1**_0_);
- **Subpartition**, subpartition for the side line of the structure, (e..g. _UP_1_**0**_);
- **Color**, color for the lines of the structure.

To change structure lines color it is necessary to click on the control **Color**. Opens dialog box **Select Color** (dr. 12).

Select required color (e.g. 30, kind of orange). After pressing **OK** dialog returns and **Color** will get new color.

In the area **Boundaries by length** there are fields defining ship structure limits (abscissae) by length:

- **Begin frame**, number of main frame to set aft boundary abscissa for structure building;
- **Addition** (of begin frame), addition from begin frame to fore, to set aft boundary abscissa for structure building; may be zero if boundary is exactly at main frame;
- **End frame**, number of main frame to set fore boundary abscissa for structure building;



Drawing 13. Color selection dialog

- **Addition** (of end frame), addition from end frame to fore, to set fore boundary abscissa for structure building; may be zero.

In the area **Trimming boundaries** there are trimming features for those lines that will be used as trimming boundaries (e.g. shell lines to trim deck beams).

In this area there are two checkboxes: **Begin** and **End.** To the right there are partition and subpartition for trimming structures:

- checkbox **Begin**, if set then structure will be trimmed at the begin part (for bulkheads);
- checkbox **End**, , if set then structure will be trimmed at the begin part (for bulkheads);
- edit box **Partition**, partition of the line for the structure used as trimmer (begin + end);
- edit box **Subpartition**, subpartition of the line for the structure used as trimmer (begin + end).

Numbers of partition and subpartition for building structure is entered by the user. They must be used only once (not to be repeated).

In the area **Intersection with aft/fore profiles** there are two checkboxes:

- **Aft profile**, serves for extending of structure sheer line up to sternframe (if checkbox is set) located on the layer from project general data (usually _**BT0_0_1**_);
- **Fore profile**, serves for extending of structure sheer line up to stemframe (if checkbox is set) located on the layer from project general data (usually it is _**BT0_0_2**_).

In the area **View or modify** there are three buttons:

- **Sheer**, go to the window for editing sheer data**;**
- **Camber**, go to editing camber data;

- **Frames**, go to the window for interpolation of intermediate (auxiliary) frame lines needed for calculation of deck knuckle lines. Often these frame lines are missing in the preliminary hull forms wireframe model (linear interpolation between two neighbor main frames is used).

Intermediate frames that are necessary for boundaries of deck zones must be inserted or interpolated in the model before building this deck. The same idea must be applied to shell butts and similar lines.

Checkbox **Save in file** of the area **Storage** rules whether to save or not structure data, created during edit actions. File name to save structure data depends on state of radio buttons:

- **Current**, gives possibility for saving data in the current revision file;
- **New revision**, creates data file with new revision number (with the same name and revision number one greater than current revision number), and stores new data (old data remain unchanged in the previous revision).

Lower row buttons are intended for:

- **Next >**, continuation of module work;
- **Cancel**, stops work;
- **Help**, view of the help topic created for this window.

Data analysis line reflects warning and error messages. Color of the square sign at the beginning shows seriousness of information (green - good, yellow - warning, red - serious error).

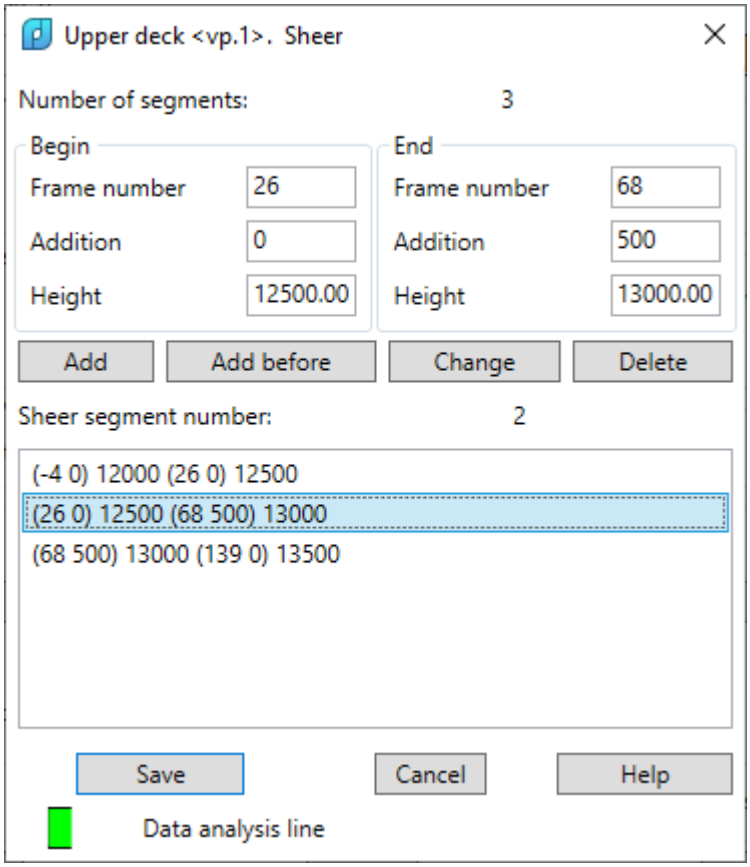
7.5. Sheer

Button **Sheer** of the structure general data dialog opens window for editing sheer line data (dr. 13). Sheer is a line in CL, used for generating longitudinal structure surface (mostly deck surface) while dynamic moving camber line along sheer, from aft to fore. Here is sample sheer window based on data for upper deck.

This dialog is intended for defining straight segments of the sheer line located in CL (center line plane, or diameter plane). Each line corresponds to a single sheer segment. On the drawing there are three segments of deck are shown. First segment starts at height 12000 mm from base line (base plane) at abscissa, described as frame -4 and addition 0, and ends at height 12500 mm at abscissa, described as frame 26 and addition 0. This means that the first sheer segment is a straight line going from frame -4 to frame 26, changing height from 12000 to 12500.

Second segment of sheer starts at h = 12500 mm at frame 26 and addition 0, and ends at h = 13000 mm at frame 68 and addition 500 mm (this is a middle plane frame). Third segment is going from h = 13000 mm (frame 68+500) to h = 13500 мм (frame 139+0).

In the area **Begin** of the window there are three fields displaying sheer data for the segment selected in the listbox:



Drawing 14. Sheer data

- **Frame number**, number of the main frame nearest to the begin (aft) boundary of the sheer segment;

- **Addition**, non-negative distance for defining abscissa of the begin boundary with respect to **Frame number**;

- **Height**, height at the begin boundary of sheer segment (relative BL).

In the area **End** there are similar three fields:

- **Frame number**, number of the main frame nearest to the end (fore) boundary of the sheer segment;

- **Addition**, non-negative distance for defining abscissa of the end boundary with respect to **Frame number**;

- **Height**, height at the end boundary of sheer segment (relative BL).

For editing sheer data select in the listbox line of the segment to be changed, then edit its data and click required button. There are the following buttons up to the listbox:

- **Add**, adding to the end of list в конец списка;

- **Add before**, adding new sheer segment before marked segment;

- **Change**, changing data of the marked segment;

- **Delete**, deleting marked segment.

Total quantity of sheer segments is watched and displayed in line **Number of segments**. Number of the selected segment is shown in the line **Sheer segment number**.

After editing old data it is necessary to save them with button **Save**. To restore old data click **Cancel**. Buttons in the lower part:

- **Save** — exit with saving data;
- **Cancel** — exit with return to previous data;
- **Help** — call help for the window.

Data analysis line is used for messages on errors or warnings. Square icon can be of three colors: green - OK, yellow - warning, red - errors.

If structure height in CL is constant or linear from the very beginning up to the end then it can be defined with one segment, if it does not intersect middle frame. Otherwise sheer must have interruption at middle abscissa.

Line **Number of segments** shows total quantity of sheer segments (on sample it is 3). Total number is unlimited. While adding or deleting segments **Number of segments** will change. Line **Sheer segment number** is empty if no line is selected in the listbox.

After clicking **Save** or **Cancel** program returns to the previous window, with general data.

7.6. Camber

Click on button **Camber** in the dialog **Upper deck. General data** opens dialog box **Upper deck. Camber** (dr. 14).

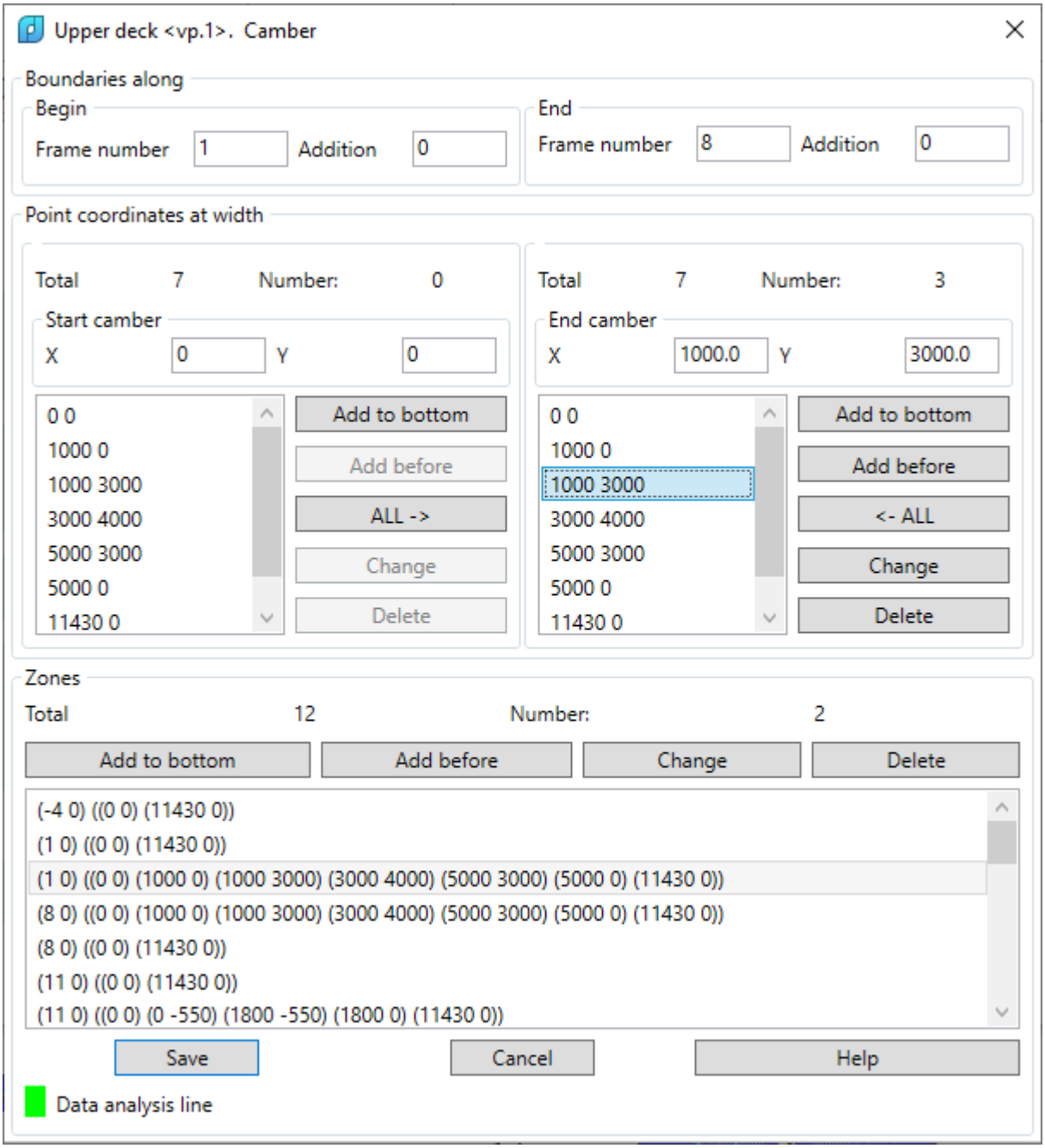
Deck camber form is defined by zones. For each zone camber line has a different equation, that develops by linear law from begin to end. Both beginning and ending cambers are defined with points to be connected with straight lines. Number of points at begin and at end must not differ more than 1. Minimal number of points in camber at any abscissa is two. Boundaries of sheer zones must be included into boundaries of camber zones.

It is usual that the first point has coordinates (0, 0). While building deck surface camber is lifted to the sheer height at the same abscissa. If the first point is not 0,0 then camber will be moved from sheer point (this is used in bulkheads). If the first camber point has nonzero coordinates (this is often for bulkheads), then all camber will be moved by X and by Y on first point coordinates values.

Between the boundaries deck surface is counted to be linear, and carcass lines are the lines between the corresponding points of camber on the aftmost and foremost abscissae. So the first point is connected with the first point, second point is connected with the second point, etc. That’s why sheer boundary points must be boundary points of camber too.

Window on the drawing is divided into three areas:

- **Boundaries along**, defining boundary abscissae of camber zone (if edited then zone line must be selected in listbox **Zones**);
- **Point coordinates at width**, points creating camber as broken line; for aft and fore boundaries;



Drawing 14. Camber data

- **Zones**, to display cambers for zones and for selection cambers for editing; two lines correspond to each zone (start camber and end camber).

Area **Boundaries along** has two subareas: for zone begin abscissa and for zone end abscissa. These abscissae define utmost boundaries of camber equation along ship (ship X axis is Z axis of world coordinate system in nanoCAD).

Area **Begin** has fields:

- **Frame number**, number of the main frame nearest to the begin (aft) boundary of the camber;

- **Addition**, non-negative distance for defining abscissa of the begin boundary with respect to **Frame number**.

Area **End** has similar fields but for end boundary:

- **Frame number**, number of the main frame nearest to the end (aft) boundary of the camber;

- **Addition**, non-negative distance for defining abscissa of the end boundary with respect to **Frame number**.

Area **Point coordinates at width** is used for entering or editing points that construct broken line as camber line at begin and end of zone. Field **Total** shows total quantity of camber points, and field **Number** shows number of camber point that is selected in the listbox at the moment.

Camber points must begin with the point nearest to center line (CL) and other points go towards shell (side). If deck beam line must be trimmed by the shell line then X coordinate of the last camber point must be equal to maximal ship halfbreadth at middle in general ship data (see file *project.par*), otherwise camber will not be cut by the shell frame polyline.

Entering points of start and end camber is made with the following principle. Fields **X** and **Y** are used for coordinates either of a new point added to camber line or of edited existing point. For editing point must be preselected in the listbox with points.

There are vertically positioned buttons designed for work with points:

- **Add to bottom**, adds point with **X** and **Y** values to the end of point list;
- **Add before**, adds point with **X** and **Y** values before the selected point;
- **ALL - >**, copies all the points of start camber to the list of points of end camber;
- **<- ALL**, copies all the points of end camber to the list of points of start camber;
- **Change**, replaces point selected in the list to point with **X** and **Y** values;
- **Delete**, deletes selected point from the list.

Quantity of input points and number of selected point are displayed in the fields **Total** and **Number** of the area **Point coordinates at width**.

Area **Zones** is intended for storing zones data and each zone has its own camber data. One zone occupies two lines in the listbox (for start and end cambers).

Upper part of area has two info fields:

- **Total**, shows total quantity of zones;

- **Number**, shows number of zone selected in the list; to select a zone user must click on one of two lines corresponding to this zone.

There are buttons in the area **Zones**:

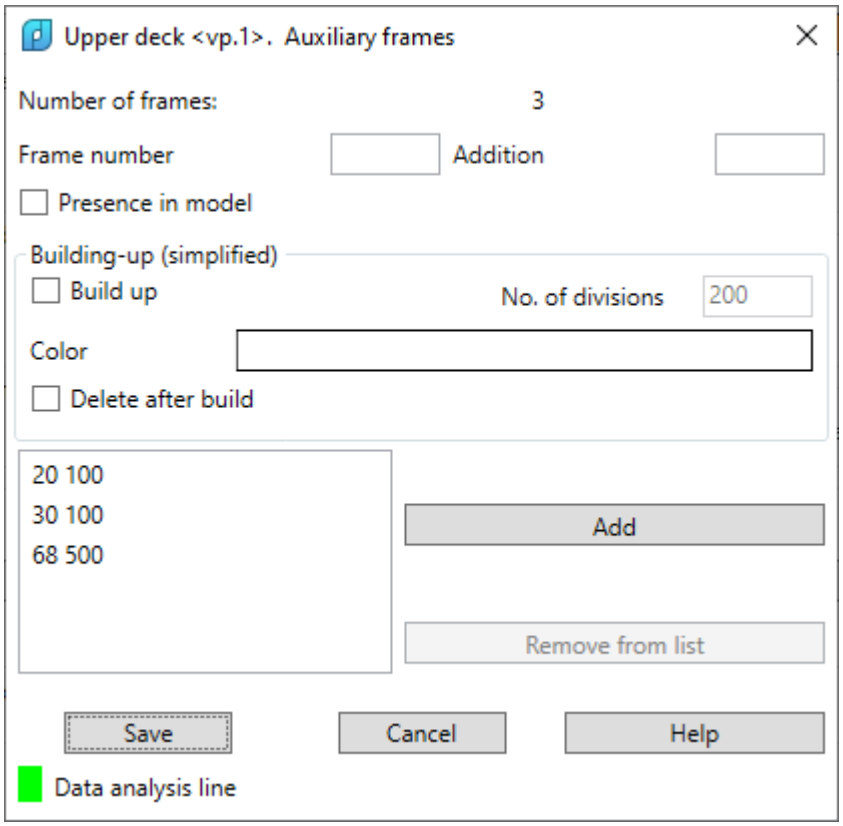
- **Add to bottom**, adds new zone to the end of zones list;
- **Add before**, adds new zone before selected zone;
- **Change**, replaces zone selected in the list to new zone;
- **Delete**, deletes selected zone (the last one cannot be deleted).

In operations of adding or replacing zone data from areas **Boundaries along** and **Point coordinates at width** are read and copied to listbox **Zones**.

For saving data of camber zones it is necessary to click button **Save**. For cancelling entered data and returning to previous saved data one should click button **Cancel**.

7.7. Auxiliary frames

Dialog box **Upper deck. Auxiliary frames** (dr. 15) is being opened if in the dialog **Upper deck. General data** button **Frames** was clicked.



Drawing 15. Auxiliary frames data

This window is intended for declaring all the auxiliary frames (i.e. frames with non-zero addition) that will be needed for structure building. Frames are named with the number of closest aft main frame and addition from main frame to fore. Frames will be built in the model in a simplified manner, by linear interpolation between two nearest frame lines to aft and fore.

Line **Number of frames** shows number of framed already added to the list of auxiliary frames. Other fields and areas:

- **Frame number**, number of the main frame closest aft;
- **Addition**, addition to fore from the main frame (addition cannot be zero);
- **Presence in model**, checking whether auxiliary frame is already present in the model. If such a frame line (two-dimensional polyline on the layer) not found then to data analysis line a message will be output, for example: *"Doesn't exist _FR55$250_0_0_"*. Deck section line will be

built but not cut by the auxiliary frame line. If checkbox is cleared (unset), then checking is not run and this beam will not be built;

- **Building-up (simplified)**, this area will be enabled if checkbox **Presence in model** is set. Allows building required intermediate frames with interpolation procedure. Estimation of possibility of application for linear interpolation is a user's responsibility (in the region of parallel body or conical surface precision is adequate but in areas of big curvature varying this method can provide not good results);

- **Build up**, creation and extension of list of frames to be built-up;

- **No. of division**, quantity of division points for two neighbor frames taking part in interpolation operation (200 is default);

- **Color**, color for auxiliary frame lines. In the **Full run** mode (if auxiliary frame is missing) a color is set that was used in the first calculation;

- **Add**, adds new frame to the list;

- **Remove from list**, removes frame from the list.

In the listbox auxiliary frames are automatically sorted by numbers (if numbers are equal then additions applied).

Buttons in the lower part:

- **Save**, saving data;

- **Cancel**, closing window without saving data;

- **Help**, viewing help topic for this window.

Data analysis line is used for messages and warnings (with square colored icon). Icon changes its as in previous dialogs.

7.8. Service window

In **Data** mode (selective calculation) after click on **Next >** button in the window **Upper deck.**.**General data** there opens dialog box **Upper deck. Service** (dr. 16). It includes a series of checkboxes.

In the upper part there are two radio buttons:

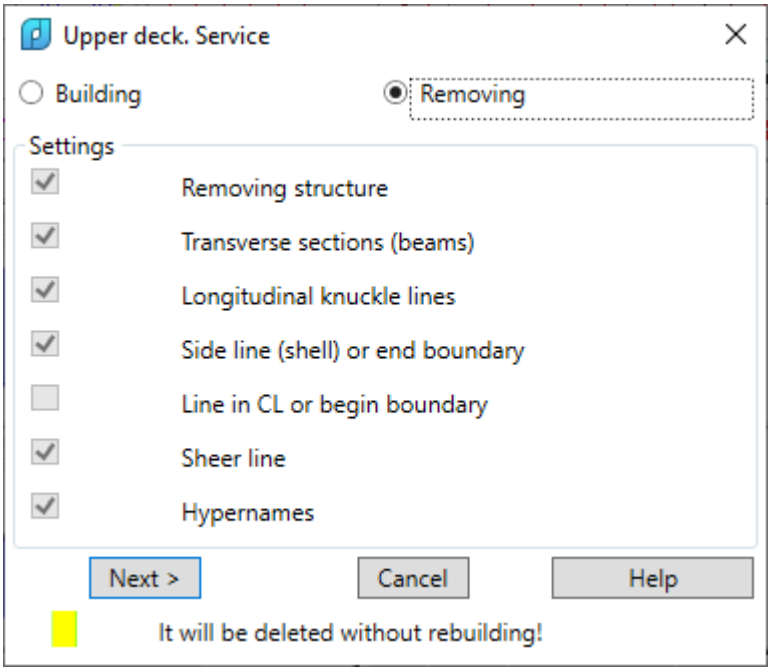
- **Building**, building structure with prepared data,

- **Removing**, deleting existing structure lines from the model. If radio button is selected then message in the **Data analysis line** will be shown: *"It will be deleted without rebuilding!"* and checkboxes will be disabled.

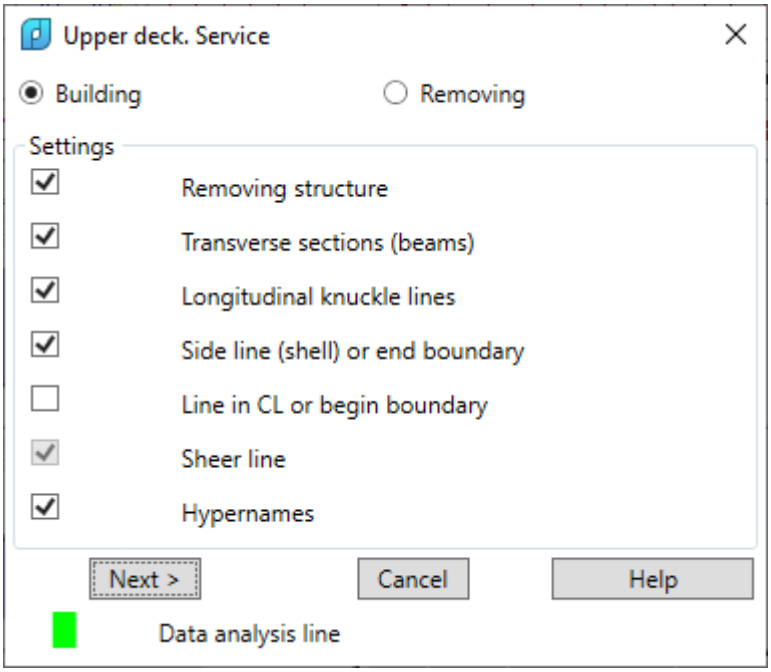
- If radio button **Building** is activated then all the checkboxes in the **Settings** area are enabled (dr. 17):

- **Removing**, previous structure lines are deleted before rebuilding.

- **Polygonal mesh**, creation of meshes (deck will have mesh on the structure surface zones). User will be able to run rendering options, especially in command 3DORBIT. Each mesh has its own color and user can visually differ structures.



Drawing 16. Structure service data (mode **Removing**)



Drawing 17. Structure service data (mode **Building**)

- **Transverse sections (beams)**, building beam lines (transverse section lines). Sometimes beam lines are not needed, then checkbox must be cleared and only side line will be generated (e.g. stringer attachment line on shell).

- **Hiding transverse sections (beams)**, hiding structure transverse section lines (beams), without freezing layers.

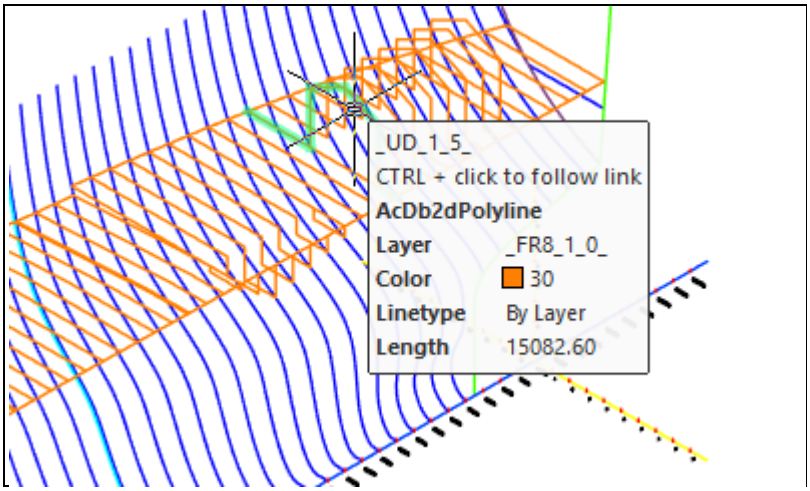
- **Longitudinal knuckle lines**, there are generated longitudinal lines connected intermediate camber points between start and end.

- **Side line (shell) or end boundary**, building structure side line.

- **Line in CL or begin boundary**, deleting other lines on sheer layer.

- **Sheer line**, building sheer line in CL. Sheer line is needed for calculation of intersections with sternframe and stemframe and writing their points to resulting table (e.g. VP_71144.tab). Table will show frame numbers, two last points coordinates, intersection with shell, as well as lengths of two last segments.

- **Hypernames**, attaching hypernames to created lines. Each line can have hypername (dr. 18), usually with structure line layer name. Hypernames are shown on hovering cursor over the line.



Drawing 18. Hypernames

If in the window **STRUCTURE Module. Project <71144>, version <0>** mode **Full run** was activated, then service settings from the previous **Data** mode are used.

After view and setting checkboxes in dialog **Upper deck. Service** for going on button **Next >** must be pressed. Click on **Cancel** stops calculation.

7.9. Running calculation

Program begins structures calculation after click on button **Next** of dialog box **Upper deck. Service**. Click on **Cancel** stops calculation.

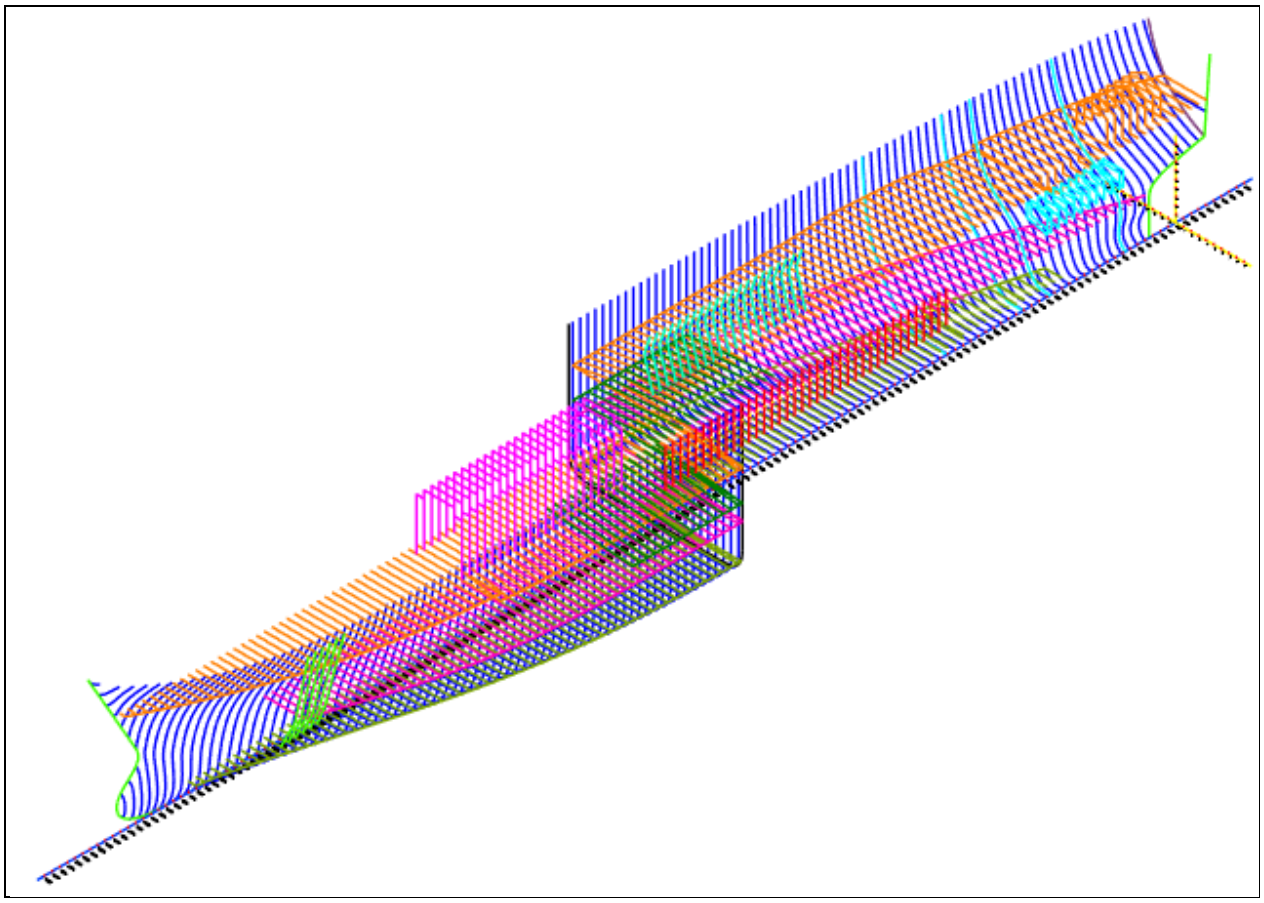
8. OUTPUT DATA

Output data can be graphical and textual.

8.1. Graphical results

Each longitudinal structure building in wireframe model is represented by a set of frame section lines (beams) and boundary (side) lines. While moving along sheer line the camber line can be constant but may be changing like ruled surface. On dr. 19 there is a model where in one session the following structures were built: main deck, lower deck, double bottom, platform and four longitudinal bulkheads, including:

- superstructure defined as bulkhead and trimmed by main deck;
- bulkhead between main deck and double bottom;
- bulkhead between main deck and lower deck;
- bulkhead between lower deck and double bottom.



Drawing 19. Built structures

8.2. Textual results

Module **Structure** forms several text files. In subfolder *Projects\71144\ STEEL\TAB,* created by module in the first session there are written files with calculation results (e.g. *VP_71144.tab*). Output data in files **.tab* are appended therefore these files should be cleaned from time to time.

Tables contain frame numbers, coordinates of two last points of beam line, as well as lengths of the last segments. Sample file contents is shown in the tab. 1.

Table 1

Coordinates table of platform points

Platform 9000, project - 71144

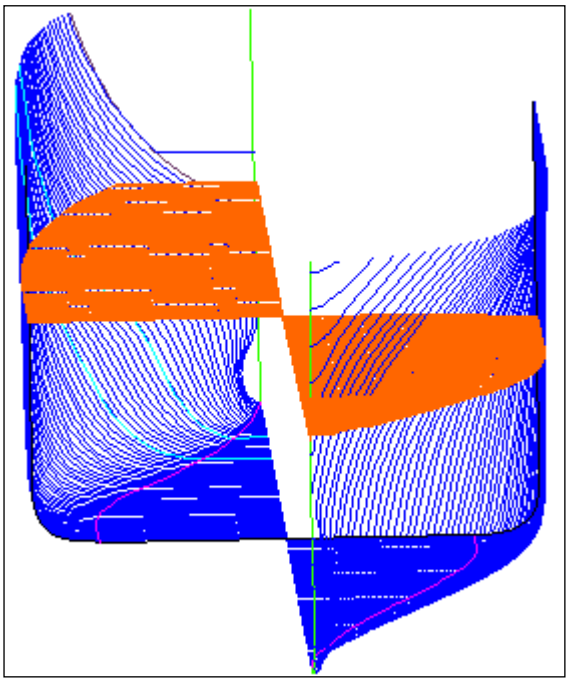
fr.	ad.	Xhp	Yhp	Xsh	Ysh	Length
57		0.00	9000.00	11428.77	8000.11	11472.42
58		0.00	9000.00	11429.53	8000.04	11473.19
59		0.00	9000.00	11429.70	8000.03	11473.36

60		0.00	9000.00	11429.64	8000.03	11473.30
61		0.00	9000.00	11429.90	8000.01	11473.56
62		0.00	9000.00	11429.95	8000.00	11473.61
63		0.00	9000.00	11429.99	8000.00	11473.65
64		0.00	9000.00	11430.00	8000.00	11473.66
65		0.00	9000.00	11430.00	8000.00	11473.66
66		0.00	9000.00	11430.00	8000.00	11473.66
67		0.00	9000.00	11430.00	8000.00	11473.66
68		0.00	9000.00	11430.00	8000.00	11473.66
68	500	0.00	9000.00	11430.00	8000.00	11473.66
68	500	0.00	9000.00	11430.00	8000.00	11473.66
69		0.00	9000.00	11430.00	8000.00	11473.66
70		0.00	9000.00	11430.00	8000.00	11473.66
71		0.00	9000.00	11430.00	8000.00	11473.66

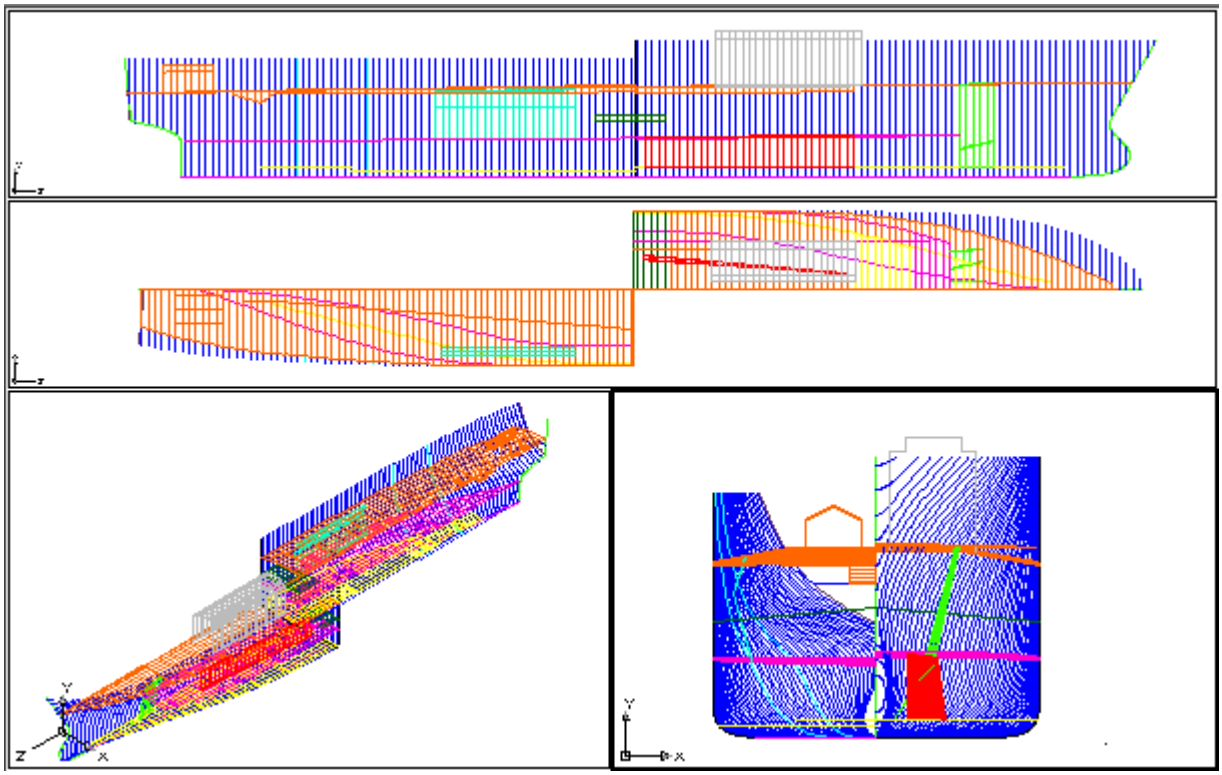
Automatically created log file is named *FF_saw_ko.$$$*, it contains module messages. New lines are appended to the file but if file size exceeds 60 Kb, then it will be cleaned.

8.3. Samples

Sample results of building structures are given in dr. 20 and 21. Visual control of results is usually run with command 3DORBIT.



Drawing 20. Sample building only one structure



Drawing 21. Sample results of mode **Full run**

On dr. 20 there is a deck built in mode **Data**, and on dr. 21 there are all the structures described in the names file *steel_<nn>.dat* and built in mode **Full run**. The same result can be achieved by selective running all these structures separately.

9. INPUT ERRORS DESCRIPTION

Module **Structure** allows to detect some obvious errors made by user while entering source data. Results of detection are shown in the **Data analysis line**, located in the lower part of dialog boxes.

At the begin of data analysis line there is a rectangular icon (“Light”), its color signals user on severity level of the error. Red light means that the work with the data is impossible and needs error fixing. Cursor automatically returns to the field with error and the field is highlighted. After correction of data calculation may be continued.

Here are some error messages supported with red square icon at the line beginning:

- “Not a number” — found unsupported symbols in number;
- “Out of: [-32768, +32767]” — input number leaved allowed interval;
- “Empty field” — found empty field;
- “Begin fr. is greater than end fr.” — begin frame number must not be greater than end number;
- “End fr. is less than begin fr.” — similar to previous message;

- “Boundaries must not be equal ” — start and end boundaries of sheer or camber zone must not be equal;

- “Abscissa 150000 outside ship” — entered abscissa is greater than maximal allowed (outside maximum from spacing table *.&sp);

- “Height should not be less than 1000” — in this situation height less than 1000 mm is not allowed;

- “Cannot be negative” — found number that must not be negative (probably for addition of auxiliary frames);

- “Zero addition” — addition of auxiliary frame cannot be zero;

- “Height should not be greater than 39000” — height is greater than twice as height (“H” = 19,5) for the project.

Messages with yellow square icon of “Light” does not prevent work continuation. User may edit some data or go on working.

Here are some yellow icon messages:

- “Initialized by zero” — value was not entered, program replaced by zero;

- ”Duplication” — repeated data;

- “Fr. 160. Initialized by 150 (max)” or “Fr.-14. Initialized by -10 (min)” — frames are extremely small or extremely big, replaced by utmost values from spacing table *.&sp;

- “Changed number -10 by -5; addition 5000 by 0“ — addition 5000 is greater than spacing value in this region. Recalculated frame number and addition according to data from spacing table.

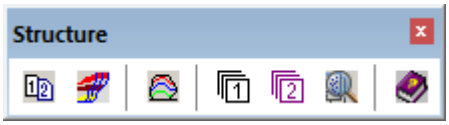
10. CALCULATION OF SHELL EXPANSION DRAWING LINES

For three dimensional model with built structures it is possible to run calculation of side lines for decks, platforms, girders, shell seams and butts for future shell expansion drawing. These draft lines are being built in the same drawing with model but moved to a good height (default is 50000 mm).

10.1. Prepare calculation

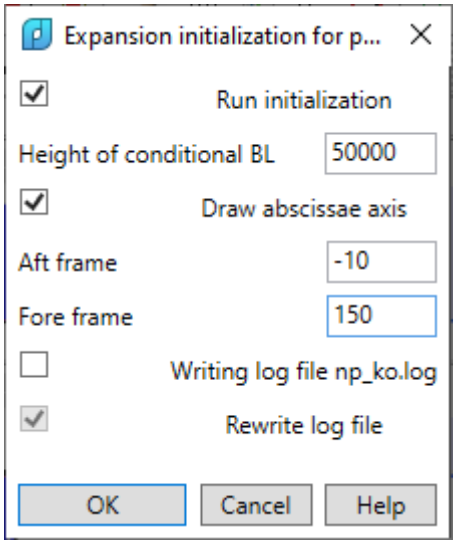
Before calling expansion programs it is necessary to open wireframe model in nanoCAD and in the dialog box **Managing projects and data versions** that is opened with menu item **STRUCTURE > Manage data versions**. User must check that the required project is active. If no then with radio button **Change current project** set right project. More on projects and data versions see hereinafter.

Expansion calculation is launched with menu item **STRUCTURE > Shell expansion** or with button  of toolbar **Structure** (dr. 22).



Drawing 22. Toolbar **Structure**

Command opens dialog box **Expansion initialization for pr.NNNNN** (NNNNN is current project name) (dr. 23).



Drawing 23. Window **Expansion initialization for pr.**

After running initialization or cancelling it (previous or default data will be applied) user will get request for selecting lines that are to be expanded. After selection press **Enter** and these lines will be generated.

10.2. Initialization

Dialog box **Expansion initialization** allows user to enter initialization parameters. For running initialization procedure it is necessary to check box **Run initialization** (otherwise parameters will be taken from the previous session or by default). Initialization parameters are the following:

- height of conditional base line (BL) relative to which the expanded structure lines will be drawn;
- checkbox for drawing axis line on the height of conditional BL;
- frame numbers, limiting drawn conditional BL;
- checkbox of using log file np_ko.log that is created in folder *Steel* of current project and is used for module calculation messages.

Parameters will be saved and suggested in the next session of expansion calculation. After exiting nanoCAD parameters in memory will be lost.

Setting checkbox **Run initialization** gives opportunity to define or change the height of the conditional base line (height is entered in the field **Height of conditional BL**). Height value may be either positive or negative. It must guarantee such a location when expanded lines do not overlap

main model lines (so they must be located upper or lower). The default value for conditional BL height is 50000 mm.

Setting checkbox **Draw abscissae axis** ensures drafting axis line on the height of conditional base line. Line limits are defined by frame numbers (fields **Aft frame** and **Fore frame**). Line is created on layer _BL with color 7 (black/white).

10.3. Calculation

After closing dialog box **Expansion initialization** with **OK** module generates request: "*Select lines to be expanded:*". User must select frames lines and longitudinal attachment lines of these structures to be expanded. Selected lines must be polylines with vertices on model frame abscissae.

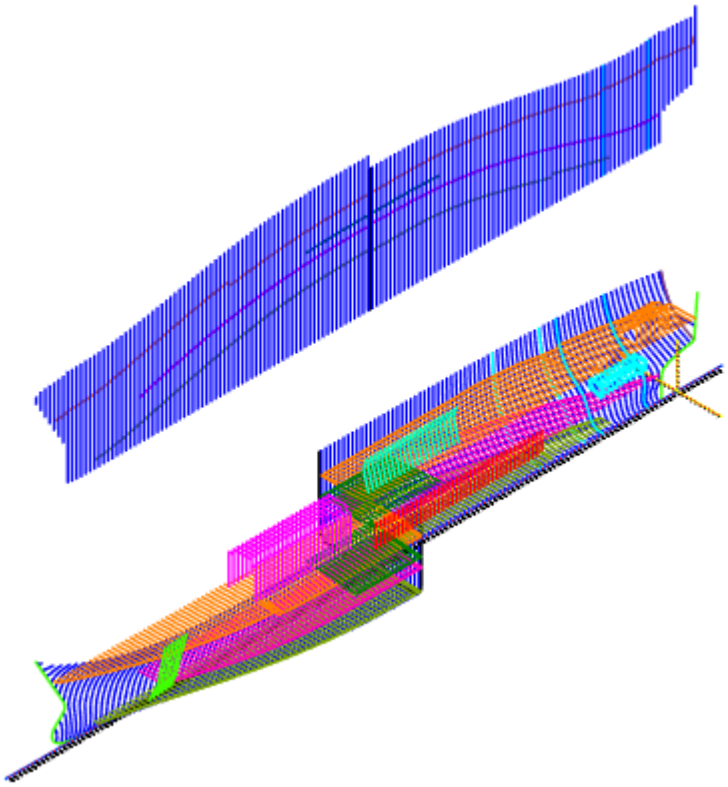
For every vertex v of expanded line program selects corresponding frame line in the model (on the layer with partition 0 and subpartition 0) and calculates length along frame line up to the vertex point v. Received value is recalculated relative conditional BL. If the frame line is uncomplete (for example, frame is divided into parts), then it can arise a downfall on the expansion drawing. It is necessary either delete received point or to join frame parts into one line adding CL segment and recalculate expanded line.

Vertices that are on the abscissae that do not correspond any model frame lines are ignored (with sample message*: Frame _FR15$276_0_0_ not found*).

Expanded lines are built as 3D polylines (broken lines). If necessary they are later to be fitted with spline tools. Color and layer of expanded lines are inherited from their parent objects in the model (with one modification: layer name of the expanded line has one more subscription symbol "_" as additional prefix). Dr. 24 shows sample result of calculating expanded side lines for decks, platforms, seams, etc., drawn over the source 3D model.

For completing expansion drawing all the built lines (conditional BL, expanded lines) are to be moved into a separate drawing file, e.g. with command WBLOCK.

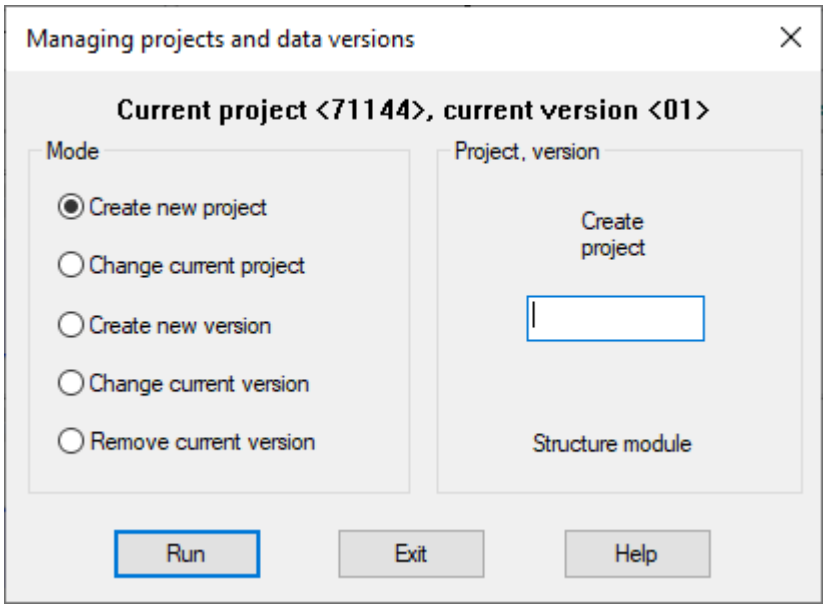
More drawing decoration (copying stern/stemframes, adding transverse stringers, creating inscriptions, etc.) is done with instruments of graphical processor.



Drawing 24. Result of expansion calculation

11. PROJECTS AND DATA VERSIONS MANAGEMENT WINDOW

Current values of project and data version can be viewed with menu item **STRUCTURE > Manage data versions** or with button  of toolbar **Structure**. This opens dialog box **Managing projects and data versions** (dr. 25), which reflects project name and data version number used by module.



Drawing 25. Dialog box **Managing projects and data versions**

Window has two areas. In the area **Mode** there are five radio buttons: **Create new project**, **Change current project**, **Create new version**, **Change current version**, **Remove current version**.

Area **Project, version** changes its view depending on active radio button in the area **Mode**.

11.1. Create new project

Creation of new project is an operation for creation of folder with project name and creation of new files of general parameters and spacing tables. It can be executed in the dialog **Managing projects and data versions**. Other command with the same functions is menu item **MODEL > New project** (without creation of module **Structure** files).

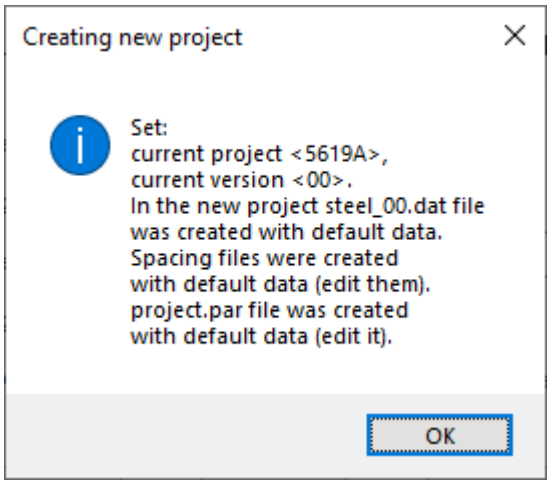
To *create new project* it is necessary in the dialog box **Managing projects and data versions**:

- in area **Mode** to activate radio button **Create new project**;

- in the field **Create project** of the area **Project, version** to enter new project name (up to eight symbols, only digits and latin letters);

- to click button **Run**.

If operation is successful then there will be a message (dr. 26) with recommendations to edit data.



Drawing 26. Message about successful creation of new project folder

After creation of new project folder it is obligatory to edit files of general parameters and spacing table files, as they are filled by standard initial data.

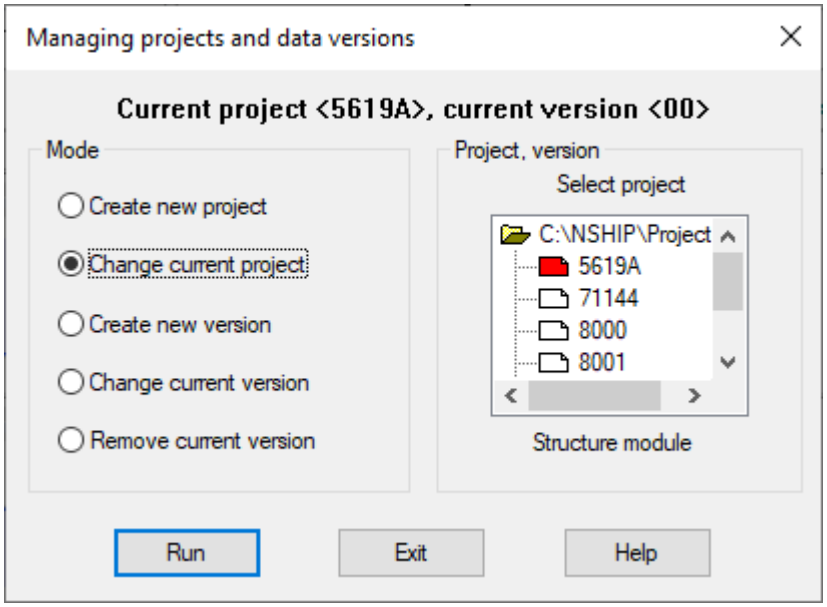
11.2. Change project

To *change project* it is necessary in the dialog box **Managing projects and data versions**:

- in area **Mode** to activate radio button **Change current project**;

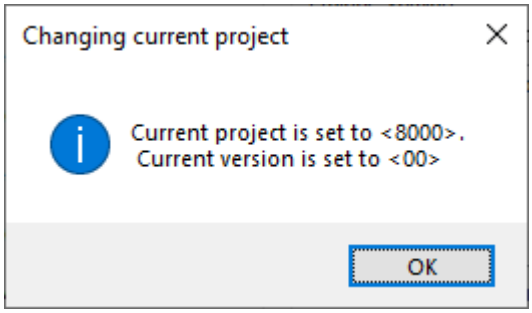
- in projects tree in area **Project, version** (dr. 27) to select project to be activated, its icon must change color to red;

- to click on button **Run**.



Drawing 27. Changing current project

After clicking button **Run** there appears message with info on setting other current project and current data version (dr. 28). In new current project version 00 is automatically activated.



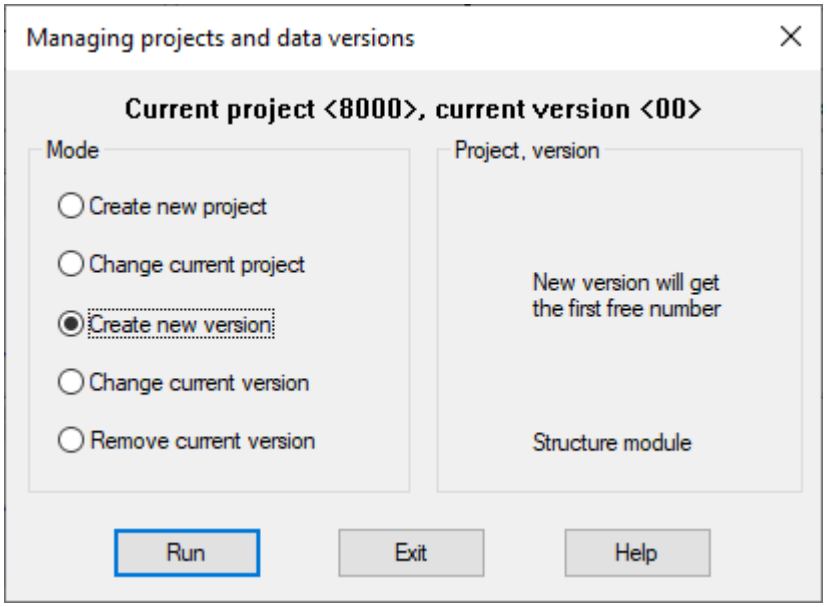
Drawing 28. Message about successful changing project

11.3. Create new data version

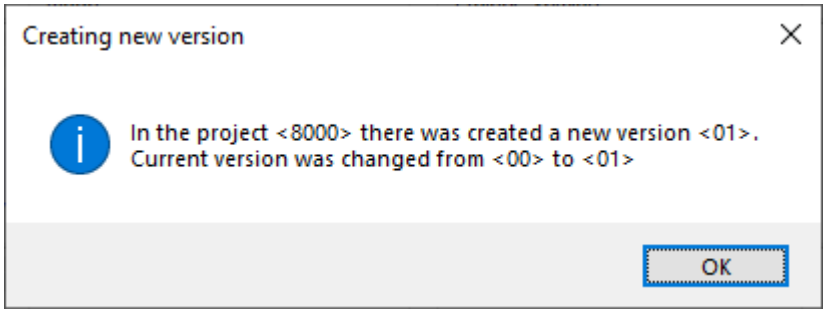
To *create new data version* it is necessary in the dialog box **Managing projects and data versions** to do the following:

- in area **Mode** to activate radio button **Create new version** (dr. 29);
- to click button **Run**.

After clicking button **Run** new data version is created for the current project (on base of current data version) and message about creation is displayed (dr. 30). As number of new version it is taken the first free number in the scope 01-99. Created version becomes current.



Drawing 29. Creating new data version



Drawing 30. Message about creation of new data version

11.4. Change data version

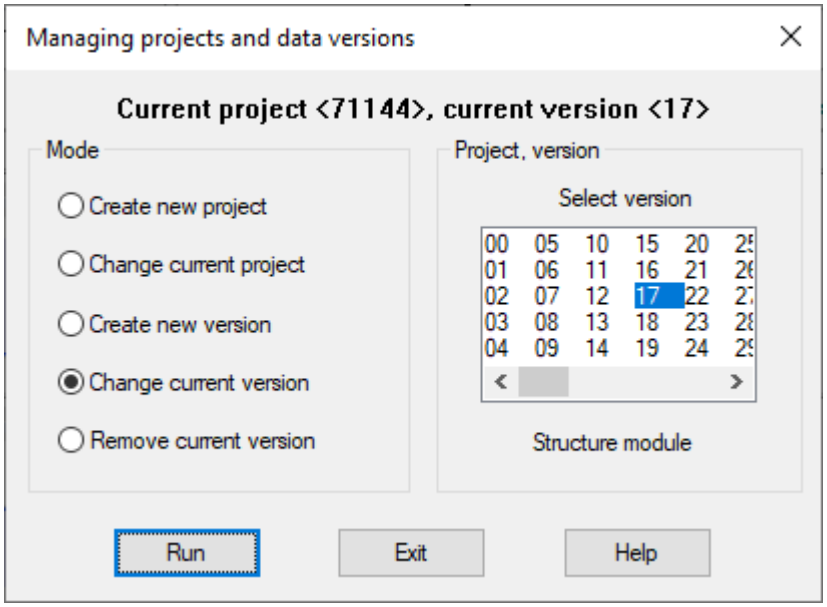
To *change current data version* it is necessary in the dialog box **Managing projects and data versions** to do:

- in the area **Mode** to activate radio button **Change current version**;

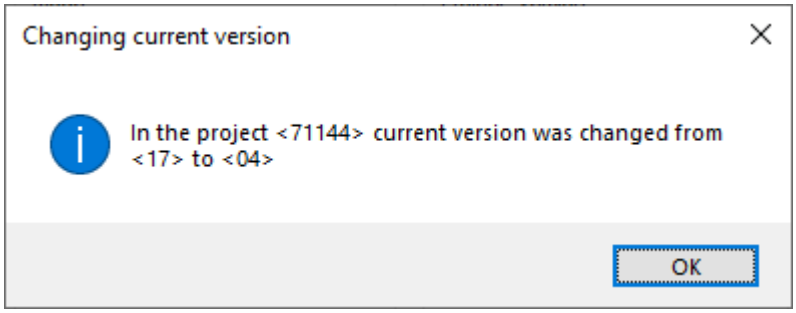
- in the area **Project, version** (dr. 31) to select number of the version that must become current (dr. 31);

- to click button **Run**.

After selecting version number and clicking **Run** there appears a message (dr. 32).



Drawing 31. Changing current data version



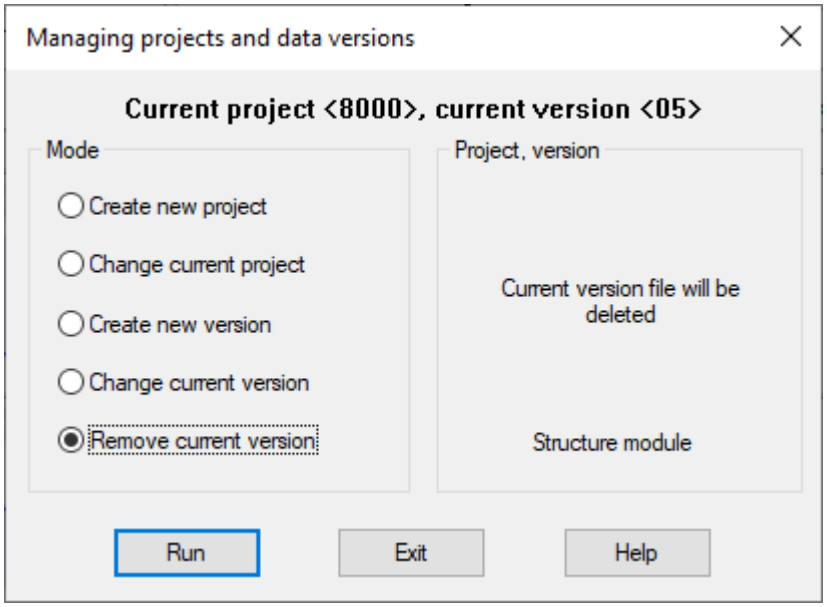
Drawing 32. Message about changing current version

11.5. Delete current data version

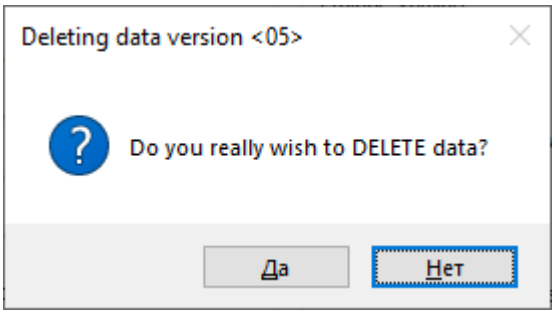
Any data version (except base version) may be deleted. First of all to *delete data version* user must set it as current. If necessary to delete it then do the following:

- set this data version file as current;
- in the area **Mode** click on radio button **Remove current version** (dr. 33);
- click button **Run**.

Deleting is rather dangerous operation, that's why program warns (dr. 34).

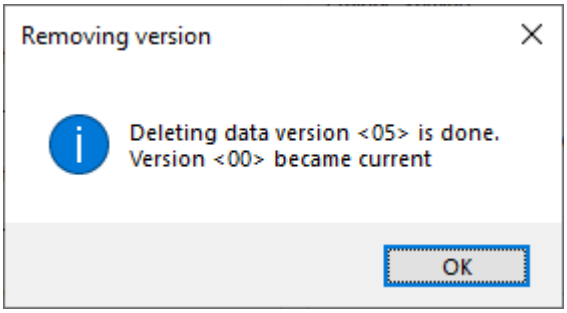


Drawing 33. Deleting current data version



Drawing 34. Warning before deleting data version file

After confirmation there is a final message (dr. 35).



Drawing 35. Message after deleting current version

After deleting, as new current version it is always set version 00 (basic). Basic version cannot be deleted.

12. LOFTBOOK CALCULATION

Loftbook is a document containing table of coordinates for sections of ship surface and structures by main planes with special spacing. Usually it is divided into two parts. Part 1 includes

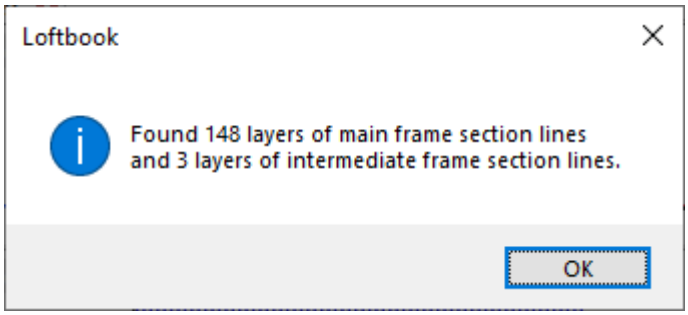
coordinates of frame lines by waterlines and by buttocks. Part 2 includes coordinates of structures (decks, platforms, stiffeners, etc.) by frames.

12.1. Start

There are three items in menu **STRUCTURE** that run commands connected with loftbook: **Loftbook, p.1**, **Loftbook, p.2**, **View loftbook**.

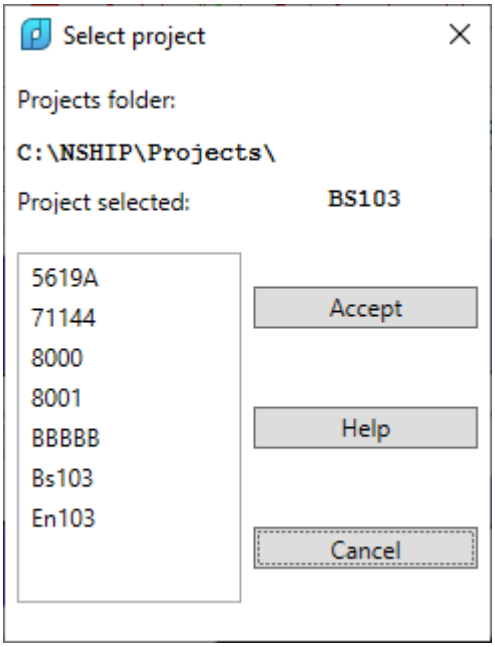
For both calculation commands it is necessary to open model file tuned to demands of the system (layer names, lines elevations) and to activate WCS (world coordinate system).

Each calculation starts with model analysis. If current drawing does not contain model lines (2D polylines on frame layers) then an error message is displayed. If model is found then there is a message with total quantity of found main and intermediate frame layers (dr. 36).



Drawing 36. Message on presence of frame layers

Next dialog box **Select project** is opened (dr. 37):



Drawing 37. Window **Select project**

In the dialog two upper lines contain info about path to the projects folder and name of current project (for module **Structure**). It is necessary to select required project in the left listbox and click button **Accept**. In case of click on **Cancel** user refuses from calculation.

12.2. Loftbook, part 1

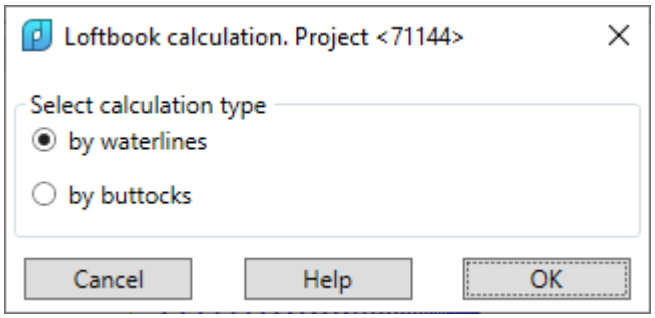
This chapter describes calculation process in the command **Loftbook, p.1** (button).

Next there is a request in command line:

User name:

User must enter any user name to be added to loftbook document (e.g. **Petrov A.L.**).

There are two variants of calculation: by waterlines and by buttocks. Then program asks to choose calculation type, in dialog box **Loftbook calculation. Project <…>** (dr. 38).



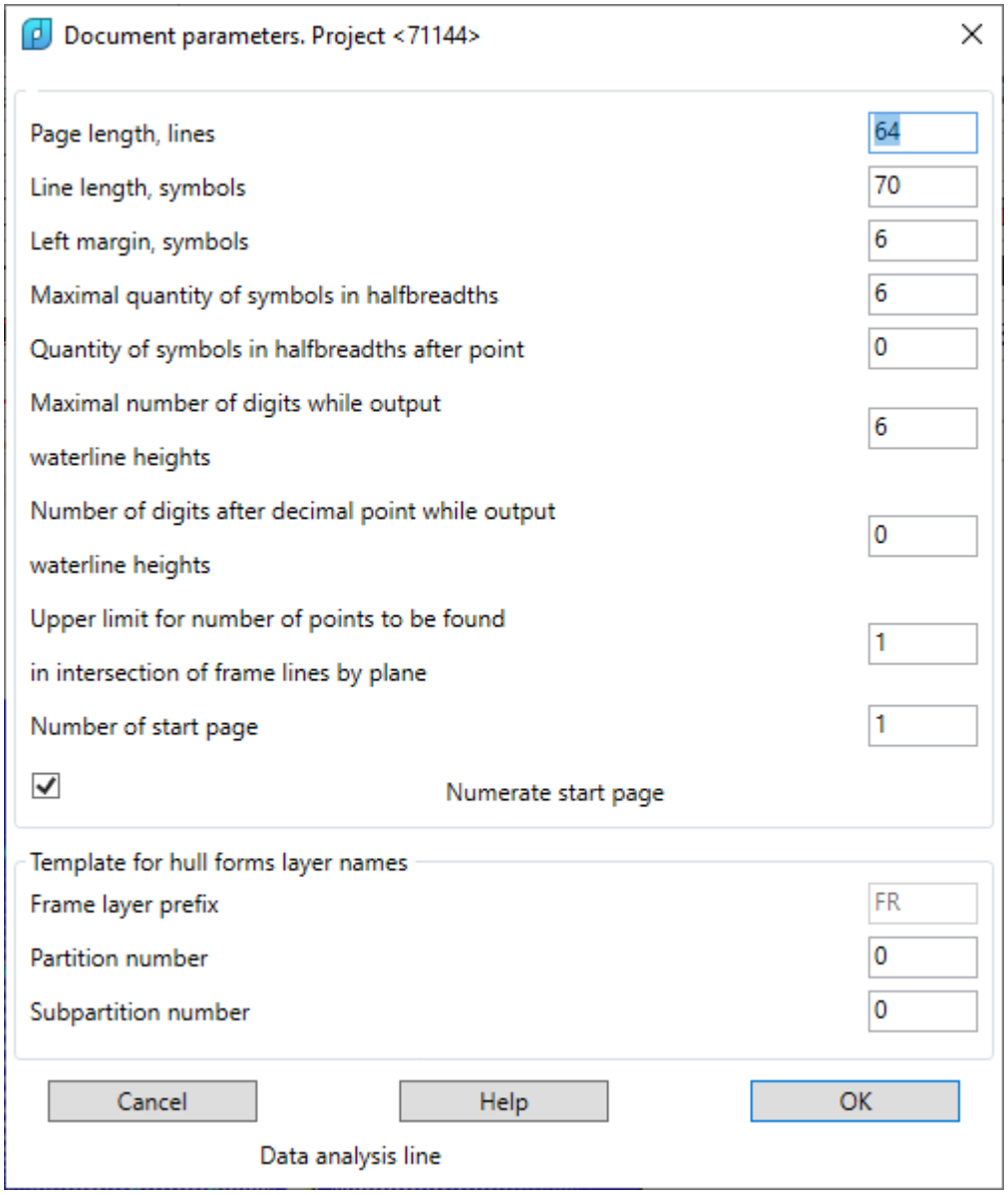
Drawing 38. Window **Loftbook calculation. Project <71144>** (mode by waterlines)

12.2.1. Calculation by waterlines

In case of selection **by waterlines** click **OK** button. Dialog box **Document parameters. Project <…>** opens (dr. 39).

It is necessary to define the following document settings for the future plain text file with fixed line length. Parameters:

- page length (9 - 64);
- line length (64 - 128);
- left margin (1 - 10);
- maximal number of digits in the halfbreadth (height) values (6 - 9);
- number of digits after decimal point in halfbreadth (height) values (1 - 3);
- upper limit of points to be searched in the operation of intersection line by the frame plane (1 - 9);
- number of start page number (1 or greater);
- partition number for shell frame lines (from -32767 to 32768);
- subpartition number for shell frame lines (from -32767 to 32768).



Drawing 39. Window **Document parameters. Project <71144>**

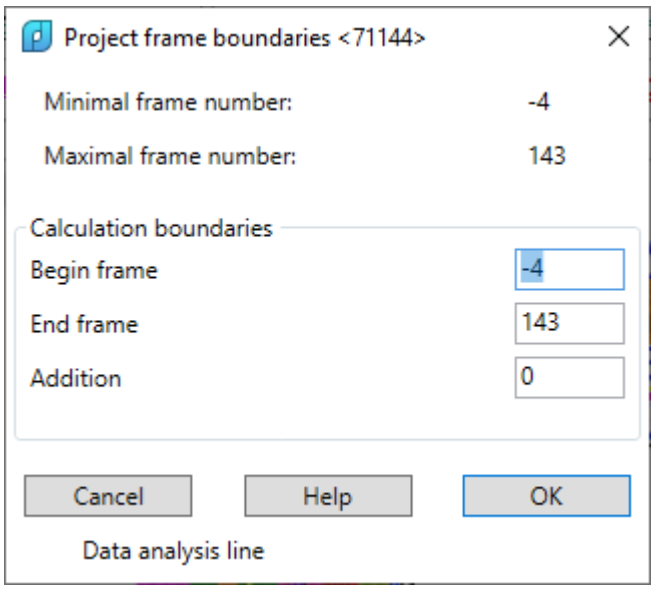
If user enters unallowed value then in the **Data analysis line** there will be an error message, and cursor will be moved to the field with an error.

Click **OK** to accept changes in parameters.

Window **Project frame boundaries <…>** (dr. 40) opens.

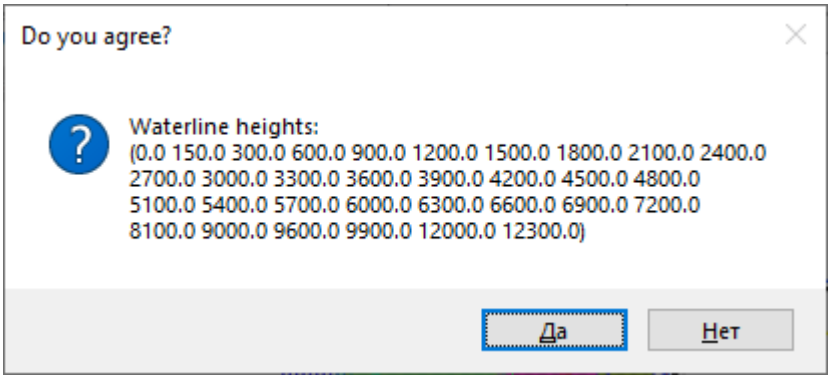
This window shows minimal and maximal possible model frame numbers for calculations (in example minimal number is -1, maximal number is 143). To reduce calculation time it is recommended to make calculation boundaries closer.

Press **OK** to continue.



Drawing 40. Window **Project frame boundaries <71144>**

Next window (dr. 41) informs that by default the following waterlines grid is suggested for calculation of loftbook.

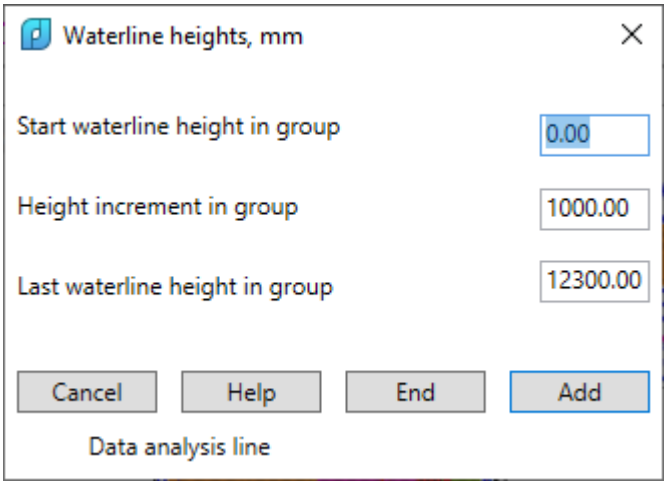


Drawing 41. Request **Do you agree?** (waterline heights)

In the initial list there are suggested 32 waterlines (0, 150, 300, …, 12000, 12300). If you wish to enter other heights then click **No (Нет)** and dialog box **Waterline heights, mm** (dr. 42) will be opened. As increment value there is suggested 1000.

Waterlines are defined in groups with constant increment between first waterline in group and the last waterline in the group. Quantity of groups can be arbitrary (at least 1). Groups are defined in height increasing sequence, from 0 mm and higher.

At first the window suggests list of heights for the first group with the following default data (final height is taken from waterlines spacing table):



Drawing 42. Window **Waterline heights, mm** (group 1 by default)

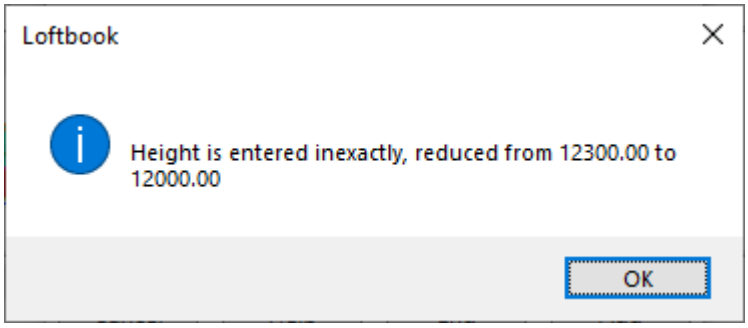
Start waterline height in group = 0.00

Height increment in group = 1000.00

Last waterline height in group = 12300.00

The last waterline height can be entered inexactly, program will check typed digits and modify end height to right value (reduce up to closest height with given increment). Therefore if to change increment on 3000, and do not edit the last height 12300, then it will be reduced up to 12000, because height increment is equal to 1000 mm.

Click on button **Add** means acception of data for the first group. Program will tell that the last height was given inexactly and will be changed to 12000 (dr. 43).

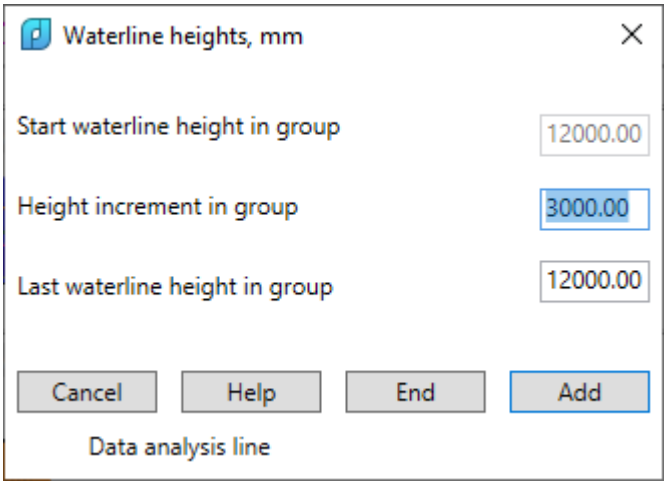


Drawing 43. Message **Height is entered inexactly**

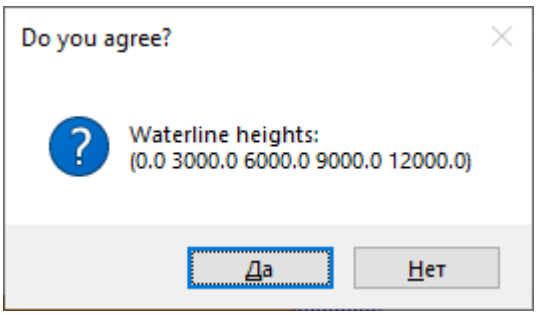
After **OK** window **Waterline heights, mm** is opened once more, with draft data for next group (dr. 44).

In this window **Start waterline height in group** is disabled and cannot be edited as it is end height for the group 1. But fields **Height increment in group** and **Last waterline height in group** are permitted for editing.

If second group is not needed then user must click button **End**. There will be request for re-newed list of waterline heights (dr. 45).

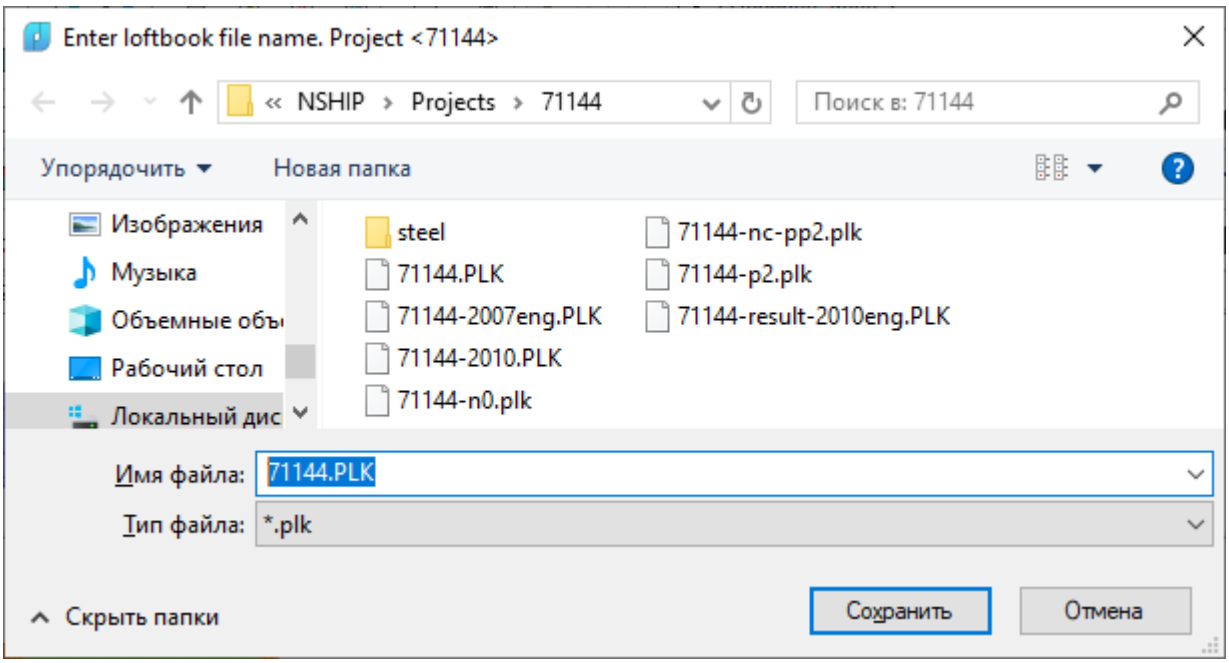


Drawing 44. Window **Waterline heights, mm** (draft data for group 2)



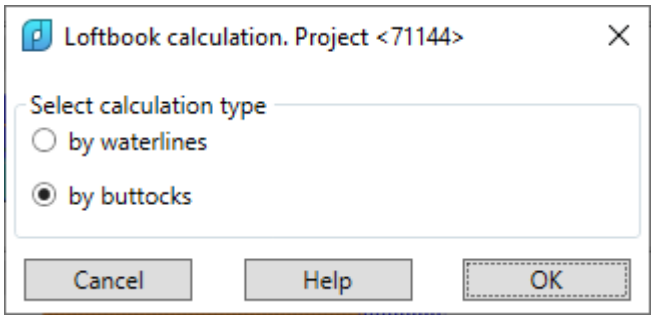
Drawing 45. Window **Do you agree?** (new waterline heights)

Confirm your approval (**Yes (Да)**). Calculation will start and at the end of it window **Enter loftbook file name. Project <…>** (dr. 46).



Drawing 46. Window **Enter loftbook file name. Project <71144>**

It is suggested to save loftbook in file with .plk extension, in the project folder. After entering name and saving, again there opens dialog box **Loftbook calculation** but radio button by buttocks will be active (dr. 47).



Drawing 47. Window **Loftbook calculation. Project <71144>** (mode by buttocks)

User can select calculation variant by waterlines but with other heights or go to calculation by buttocks. End of calculation is click on button **Cancel**.

12.2.2. Calculation by buttocks

Option of calculation by buttocks uses the same scheme as option by waterlines, but with changing waterline heights to buttock halfbreadths.

Window **Document parameters. Project <…>** for variant of calculation by buttocks is shown on dr. 48.

Decoration settings for loftbook document with fixed line length by buttocks are similar to settings used for calculation by waterlines.

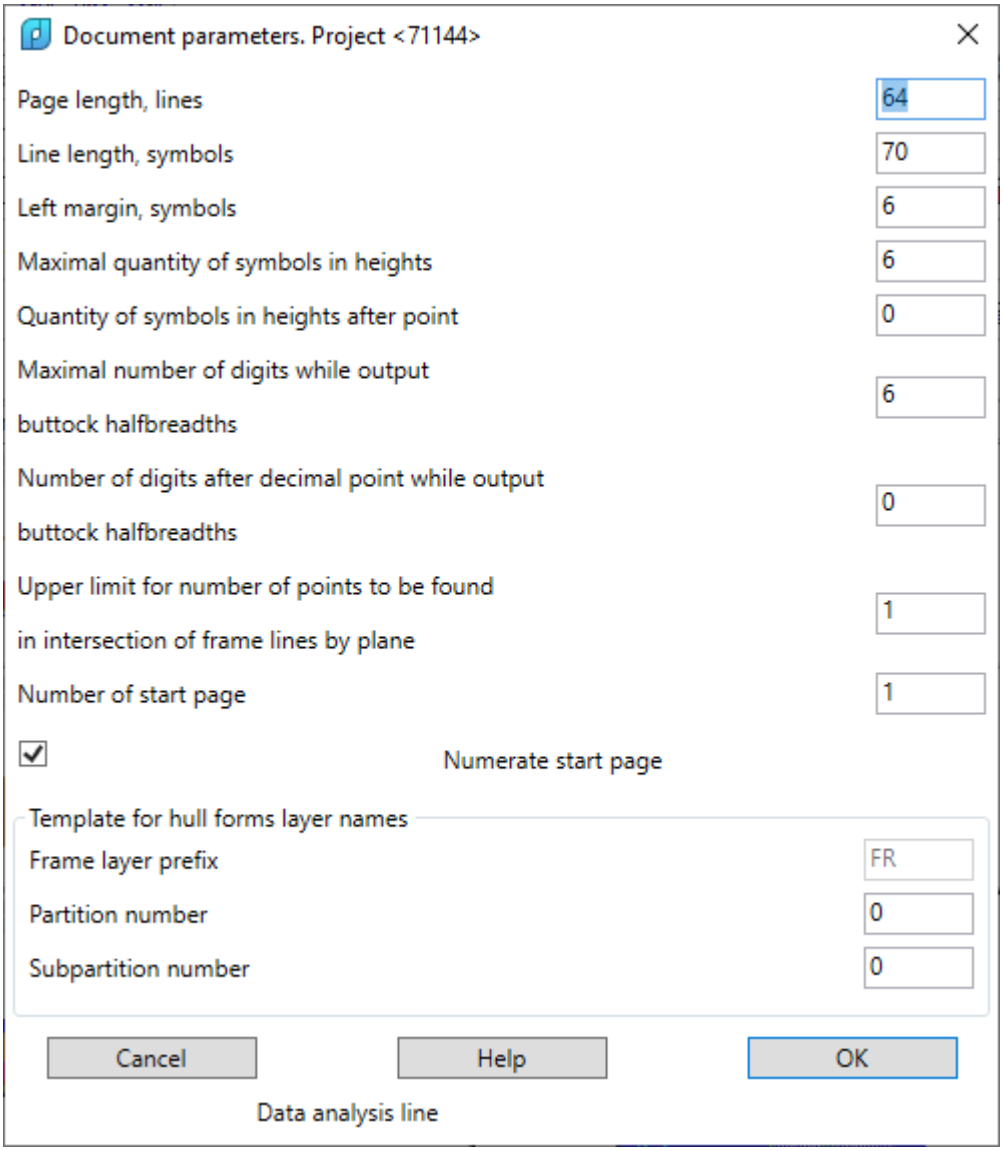
Next there is again window **Project frame boundaries <…>** (dr. 49).

Here user must also define boundaries of current calculation.

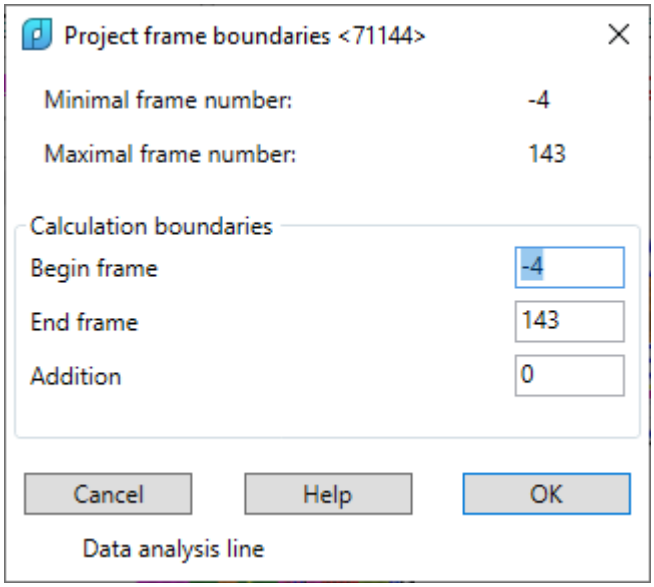
Next window (dr. 50) informs that by default the following buttock grid is suggested for loftbook calculation.

In the initial list there are ordinates of 32 Buttocks (0, 150, 300, …, 12000, 12300). To edit the list user should press **No (Нет)**. Window **Buttock halfbreadths, mm** will be opened (dr. 51). Increment is suggested with value 1000 mm.

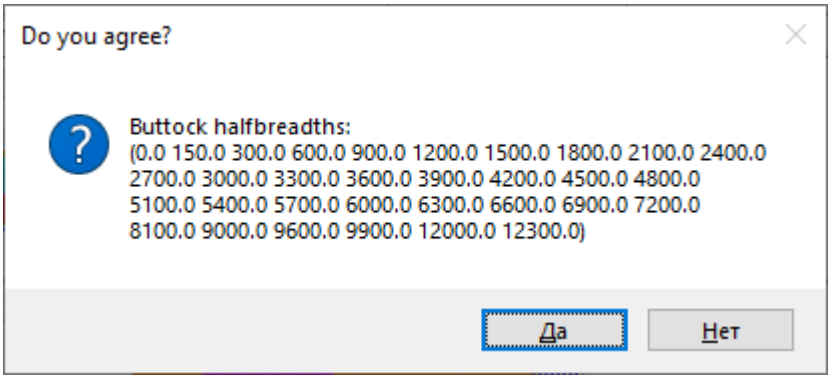
Buttocks are entered in groups with constant increment between the first and the last. Quantity of groups may be arbitrary (at least 1). Groups are given in halfbreadths increasing sequence, from 0 mm and greater.



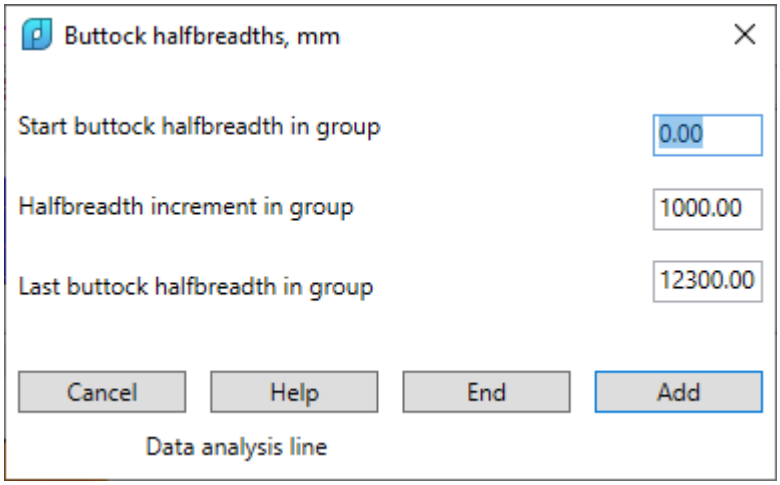
Drawing 48. Window **Document parameters. Project <71144>** (by buttocks)



Drawing 49. Window **Project frame boundaries <71144>**

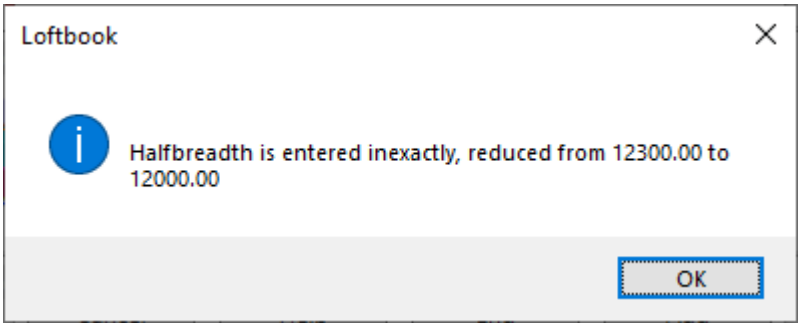


Drawing 50. Request **Do you agree?** (buttock halfbreadths)



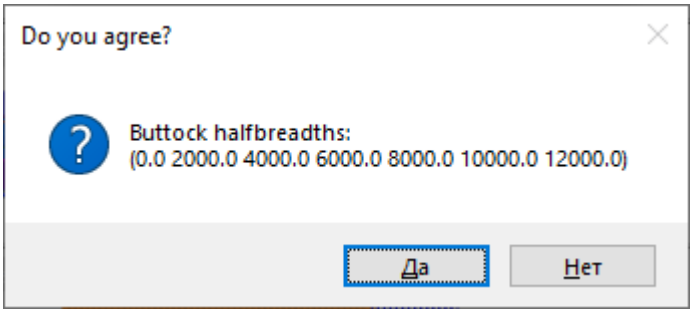
Drawing 51. Window **Buttocks halfbreadths, mm** (group 1 by default)

Pressing button **Add** will mean data approval for the first group. Program will tell that end halfbreadth is given inexactly and will be modified from 12300 to 12000 (dr. 52).



Drawing 52. Message **Halfbreadth is entered inexactly**

After click on **OK** There again will appear window **Buttock halfbreadths, mm** with draft data for the next group. If second group is needless, then press button **End**. Ther will be request for confirmation of edited buttock halfbreadths list (dr. 53).



Drawing 53. Message **Do you agree?** (new buttock halfbreadths)

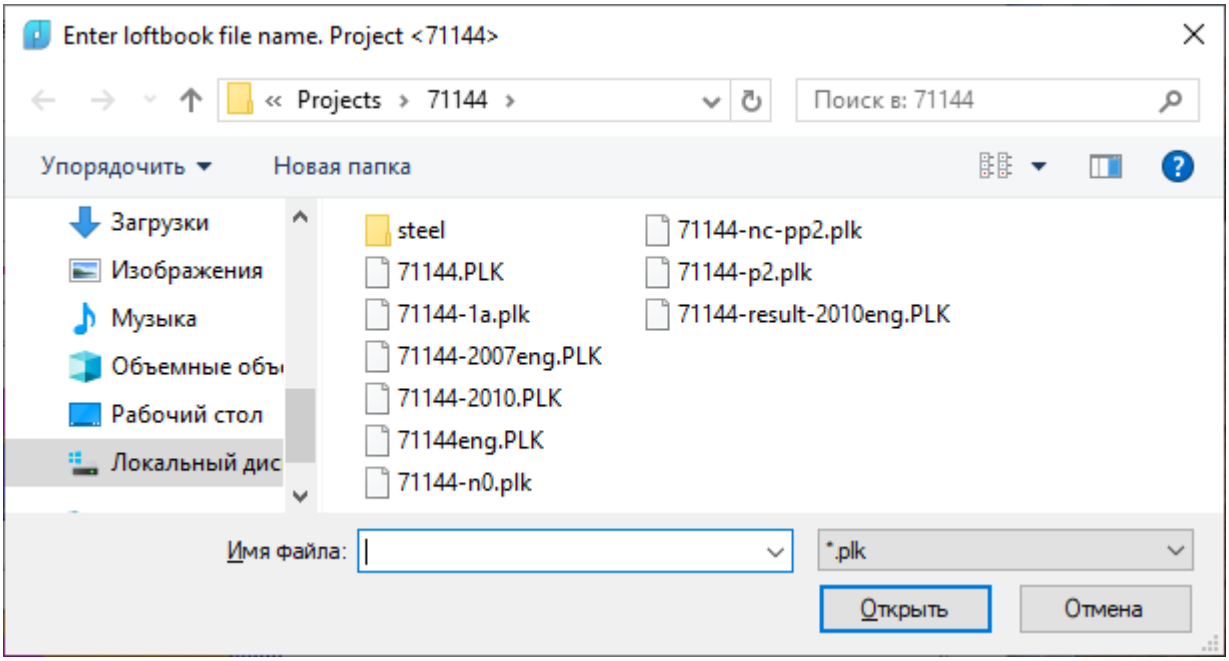
Confirm approval (**Yes (Да)**). After that calculation will be executed, data storage will be to the same file which name was typed in the first calculation (by waterlines). Next window for selection of calculation option opens once again but radio button **by waterlines** will be active.

User can continue calculation (by waterlines or by buttocks) or press button **Cancel** to stop further calculation.

12.2.3. View results

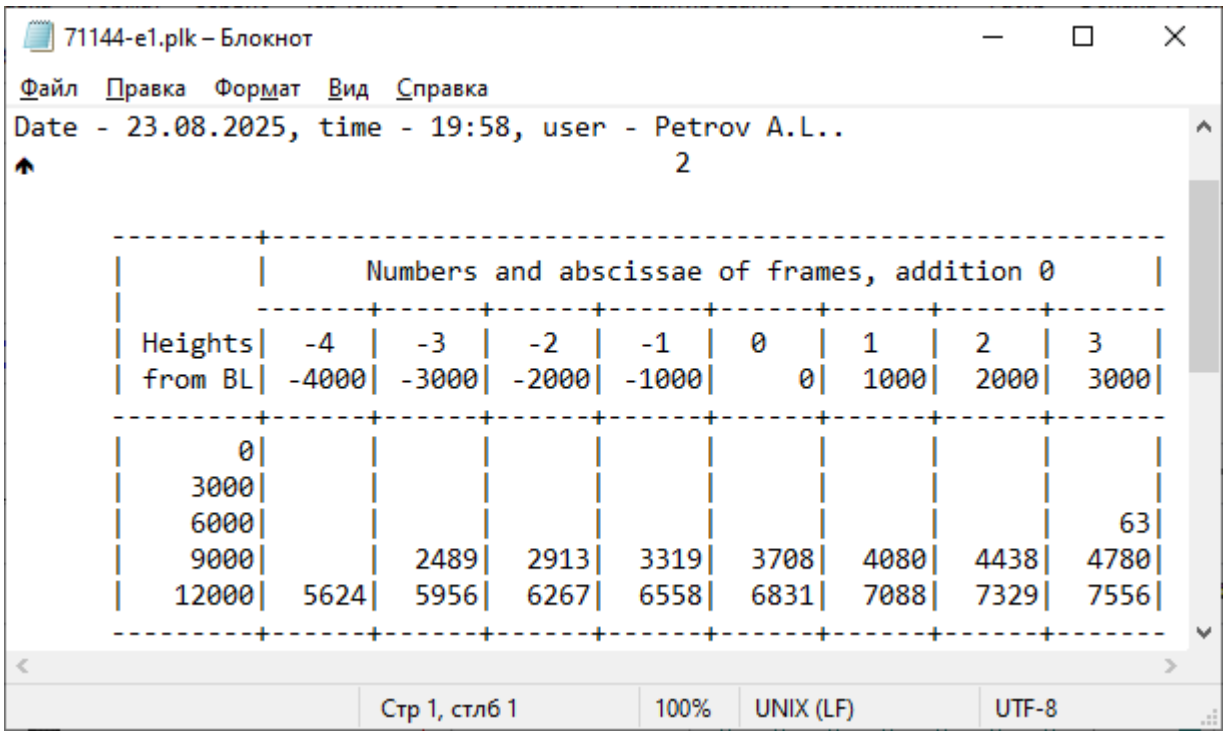
For viewing results from files with extension plk Notepad editor is used, it is called with command **STRUCTURE > View loftbook** (button).

First window **Select project** opens, in it user must choose project. Later program will suggest to select PLK file from the project folder in *NSHIP\Projects* (dr. 54).



Drawing 54. Window **Enter loftbook file name. Project <71144>**

Select file and press **Open (Открыть)**. File will be opened in Notepad (dr. 55).



Drawing 55. Viewing loftbook file

Loftbook is created as text in tabular form, using prepared settings (page size, quantity of symbols in a line, etc.).

For selected waterlines heights over BL are given, and for frames included in calculation interval, — numbers and abscissae from aft perpendicular.

Analysis of the first results page on the drawing shows that frames from -4 to 3 are not intersected by waterlines of height 0 and 3000. Frame 3 begins in CL on height between Z=3000 mm and Z=6000 mm. All the frames in numbers range (-4 3) have points on height 12 m.

12.3. Loftbook, part 2

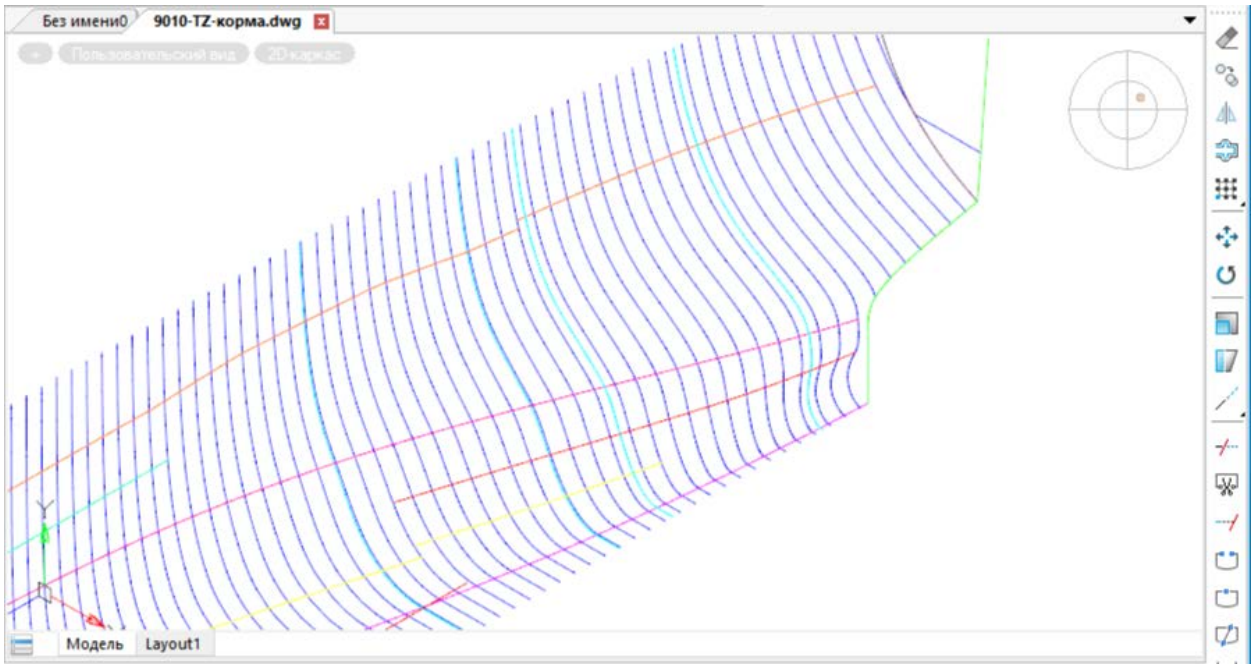
Loftbook (part 2) of system N-Ship includes coordinates of longitudinal 3D-polylines, like side lines of decks, platforms, shell stringers, etc. Lines are picked in interactive mode, for this the most convenient views are elevation, waterline or isometric (TZ). Sample model is shown on dr. 56.

For calculation user should pick deck and platform side lines of loftbook (in the sample they have light-brown, green, pink, magenta, red and yellow colors).

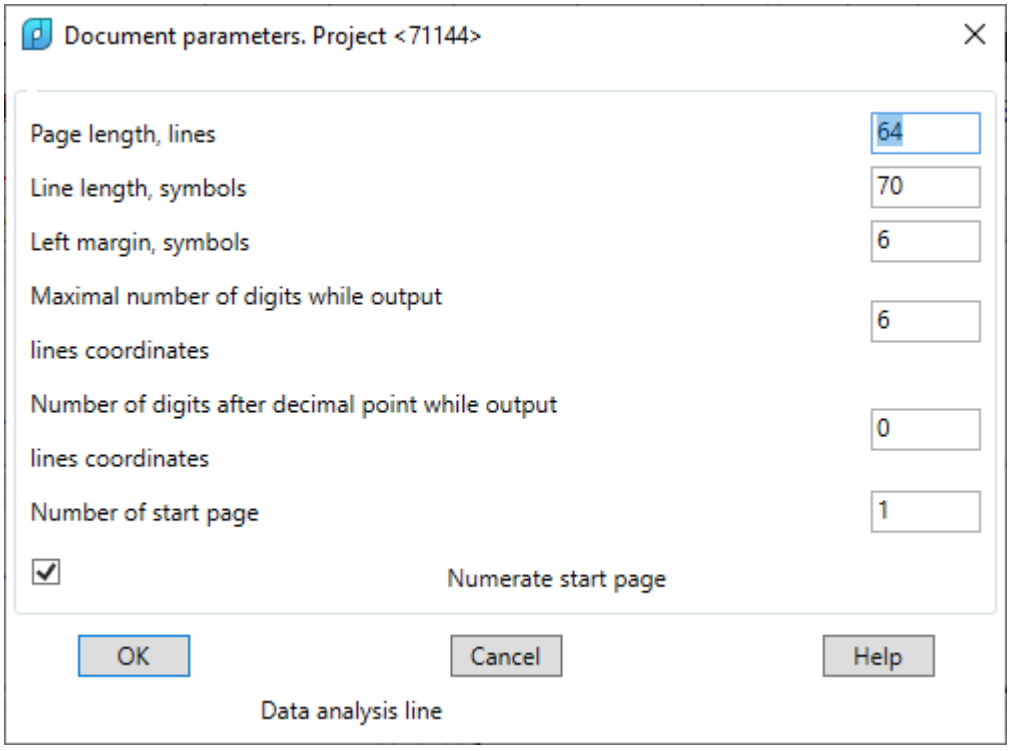
Run command **STRUCTURE > Loftbook, p.2** (button).

Program outputs message about quantity for found layers of main and intermediate frames (as for loftbook, part 1). Next in dialog box **Select project** choose project name. Reply to request in command line: *User name:*.

Next dialog box **Document settings. Project <…>** opens (dr. 57).



Drawing 56. Model for calculation of loftbook, p.2

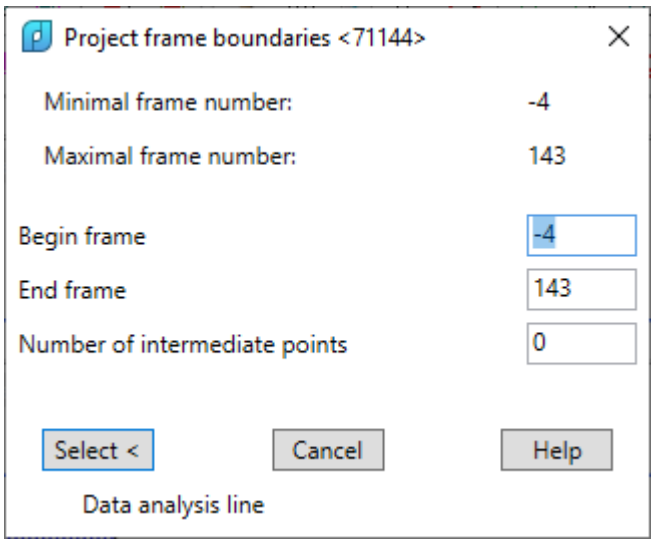


Drawing 57. Window **Document settings. Project <71144>**

In loftbook p.1 for each frame point one coordinate was output (height or halfbreadth). In part 2 for any structure (deck) line at each frame there will be two coordinates.

Enter parameter values and press **OK**.

Program opens window **Project frame boundaries <…>** in the form of loftbook p.2 (dr. 58) that is slightly different from the form for loftbook p.1.



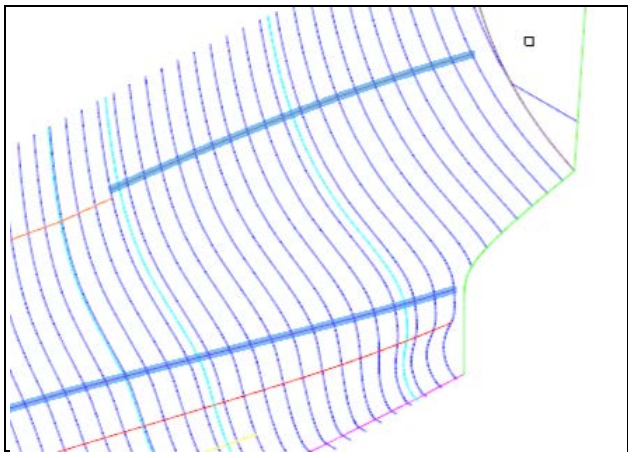
Drawing 58. Window **Project frame boundaries <71144>** (part 2)

Instead of buttons **Cancel**, **Help** and **OK** now in the window lower zone buttons **Select**, **Cancel** and **Help** appeared.

There is a new field **Number of intermediate points** (instead of field **Addition**).

Parameter **Number of intermediate points**, if its value is non-zero, allows output of structure line coordinates not only at main frames but at intermediate points too (calculated by interpolation). But user must consider that 3D-polylines usually are built on main frames and are connected with straight lines between them (simplified).

Button **Select <** realizes auxiliary function of the window: temporary exit for selecting lines. Press button and pick lines for which it is necessary to generate loftbook, p.2 (dr. 59).

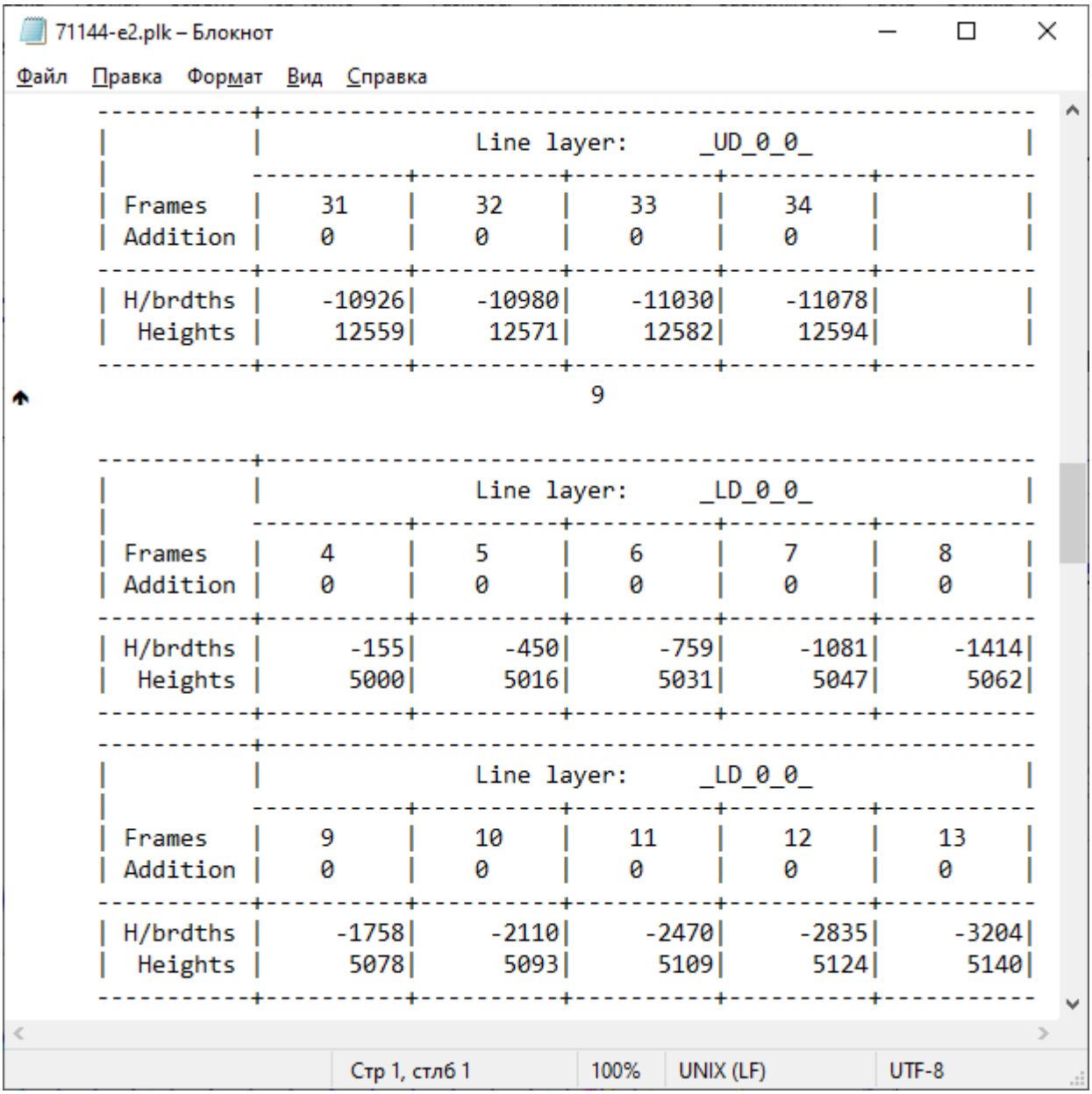


Drawing 59. Picking lines for loftbook (part 2)

After selection press Enter and window will opens again. Click button **Cancel** for finishing selection and starting calculation.

After calculation program opens window **Enter loftbook file name. Project <...>**. Enter name of the document file with .plk extension. Program will save results in it.

View this file with menu item **STRUCTURE > View loftbook**. Notepad editor will be used for viewing. For each included longitudinal line there are separate tables in PLK file of loftbook (part 2) (dr. 60).



Drawing 60. Viewing loftbook (part 2)