CAD/CAM System B-Ship+

Version 5.0

Model Structure

Creation of internal structures and superstructures

User manual

BSHIP.00009.005-2022

Sheets 41

Saint Petersburg

2022

ANNOTATION

The document is a reference manual for work with the module **Structure** of the **B-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 **B-Ship+** system for the design and technological preparation of the ship hull production and have practical experience of using BricsCAD graphical system. **B-Ship+** is informationally compatible with the systems **Ritm-Ship** (AutoCAD), **R-Ship+** (AutoCAD), **N-Ship+** (nanoCAD).

Recommended operating systems are: Windows 8.1, Windows 10.

Contact data:

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

Email: **npol50@yandex.ru**

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

Copyright © BSHIP. B-Ship+ system. Module Structure, 2016-2022. Saint Petersburg, Russian Federation.

CONTENTS

1. GENERAL INFORMATION

The described program (hereinafter module **Structure**) allows building in 3D model with prepared hull forms lines surface 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.

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

2. APPLICATION CONDITIONS

Module works in the environment of graphical editor, requires prepared and initialized ship wireframe model DWG file. Hull forms lines (frames, waterlines, buttocks) must be created in the corresponding ship UCSs. Lines of other structures (platforms, decks, etc.) taking part in building process must be 2D polylines on corresponding layers. Layer names must satisfy to B-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 (main addition=0);

<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.

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.

Note. In the last years there are many cases when wireframe model is not created inside graphical processor (BricsCAD, AutoCAD, nanoCAD, etc.) but is received as a result of DXF/DWG export from great CAD systems (Foran, Aveva, etc.). It is an important opportunity, moreover very often export can be well-prepared. First of all it is connected with frame sections of shell. Main frames (frames with integer numbers) lines are obligatory objects to be exported but many other lines (intermediate frames, butts, seams, structure sections, structure boundary lines, stern/stem frames etc.) can be ordered for getting from the export operation. The best prepared export will increase precision of building in module **Structure** and will reduce labourness.

3. USER INTERFACE

Module **Structure** needs drop-down menu **STRUCTURE** that can be loaded by command MENULOAD or menu item **BDATA > Menuload**. In the dialog box **Choose a Customization File** set type of files *.mnu (dr. 1) and select file *BSHIP\Sys\BRv19\ bship_structure_en.mnu* (on example of BricsCAD v19 and English localization of B-Ship).

.
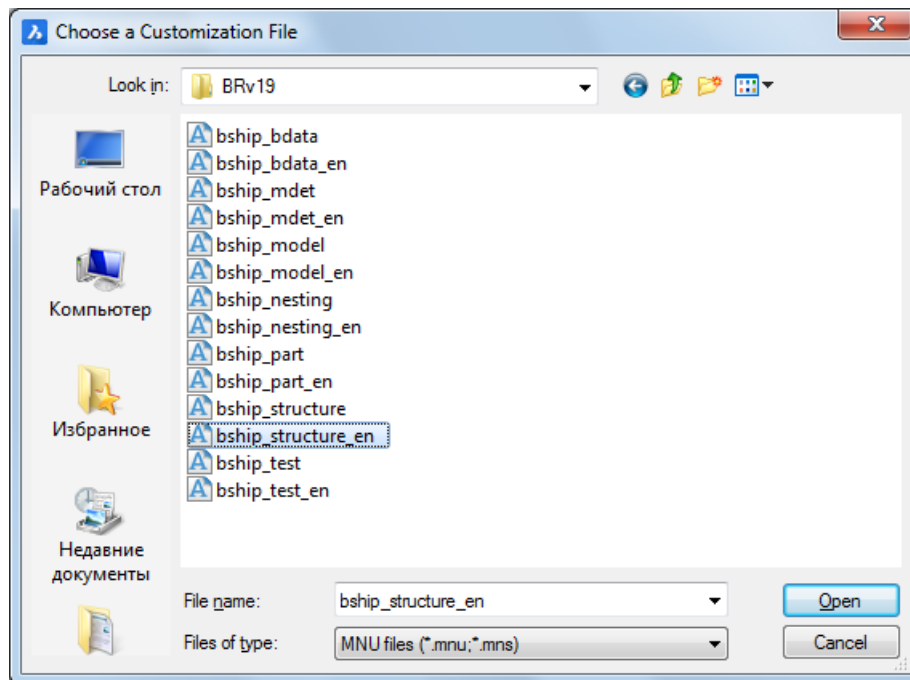
Drawing 1. Loading *bship_structure_en.mnu*

Module supports some other interface languages (e.g. Russian). To change interface language use menu item **BDATA > UI interface**.

In the menu bar all the B-Ship menus are appended with two symbol suffix defining localization, for example: **MODELru**, **STRUCTUREru** (Russian), **PARTen** (English) etc. (dr. 2). For simplicity everywhere in the document menu caption will be used without suffix (**STRUCTURE**).

Drawing 2. Menu captions

Here is menu **STRUCTURE** (dr. 3).
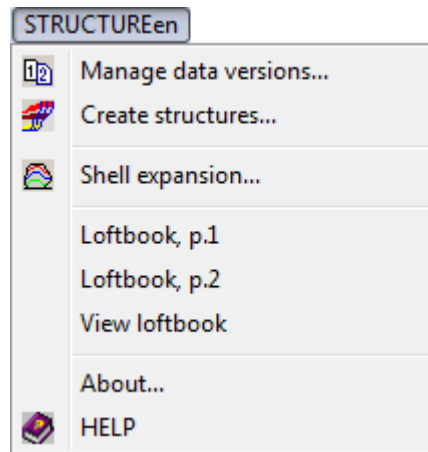
Drawing 3. Drop-down menu **STRUCTURE**

Module **Structure** functions are accessible from the toolbar **Structure** too (dr. 4).

Drawing 4. Toolbar **Structure**

4. START

For launching main calculation procedure it is necessary to have prepared structures data files (on data see hereinafter) and in the prop-down menu **STRUCTURE** click item **Create structures** (see dr. 3).

On start program verifies if there is current model DWG file 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 generated, suggesting opening model file (dr. 5). An important feature is pointing at FR prefix of frame layers (model must not be prepared in different localization).
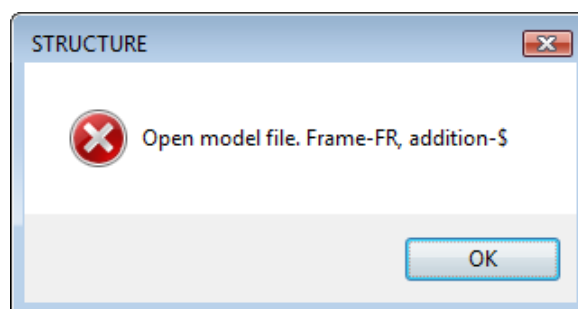
Drawing 5. Message on invalid model

Click **OK** to stop work. User must open model file with prepared hull forms and start running once more. Here is a sample 3D wireframe model on dr. 6.
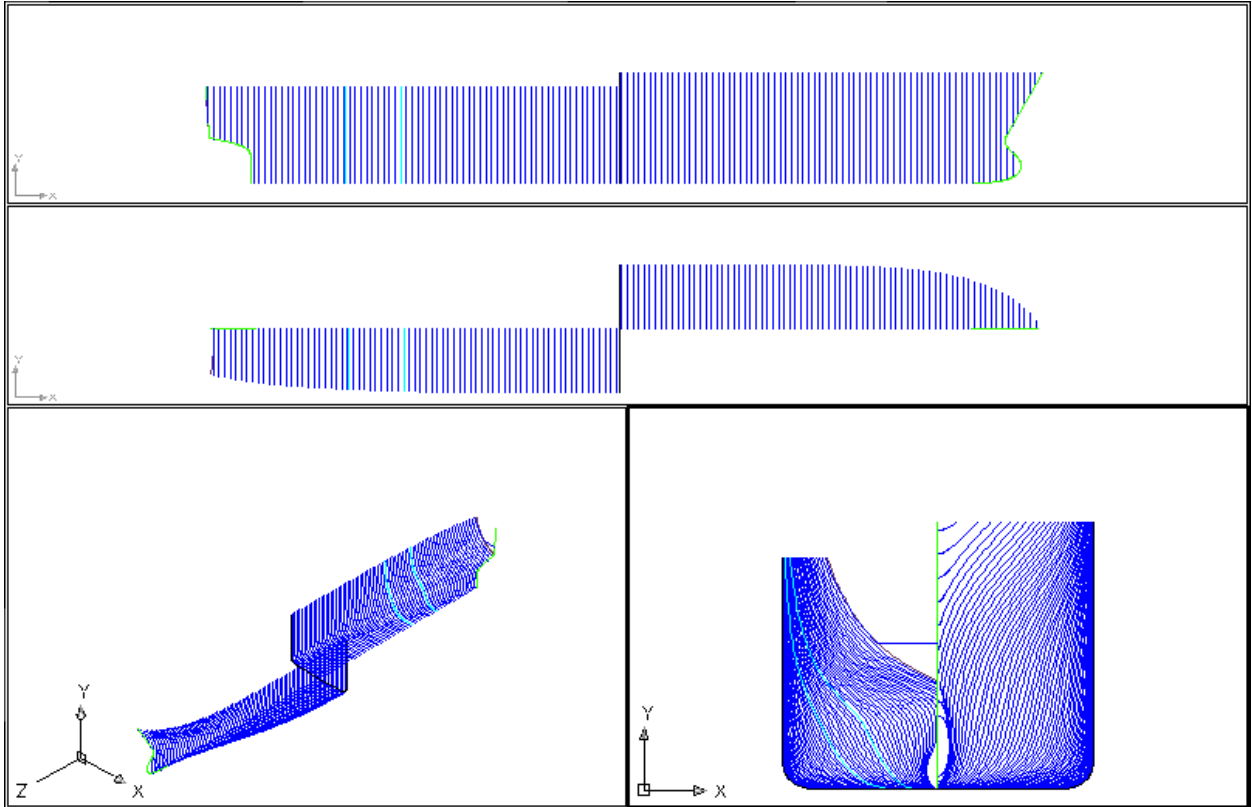
Drawing 6. Sample file with wireframe hull forms

5. INPUT DATA

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

5.1. 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:\BSHIP\ 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: \BSHIP\Projects\71144\Steel*).

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.

As in module **Model**, creating new project in module **Structure** means creating project folder and specific files needed for the module. It is done with menu item **Manage data versions**. There are created 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 71159:

General parameters file for project 71159

Bulk carrier

Design bureau

Classified

17.06.20

17.06.20

LMAX=127.00

LPP=100.00

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_*

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

LPP — length between perpendiculars, m (in the sample — 100.00);

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

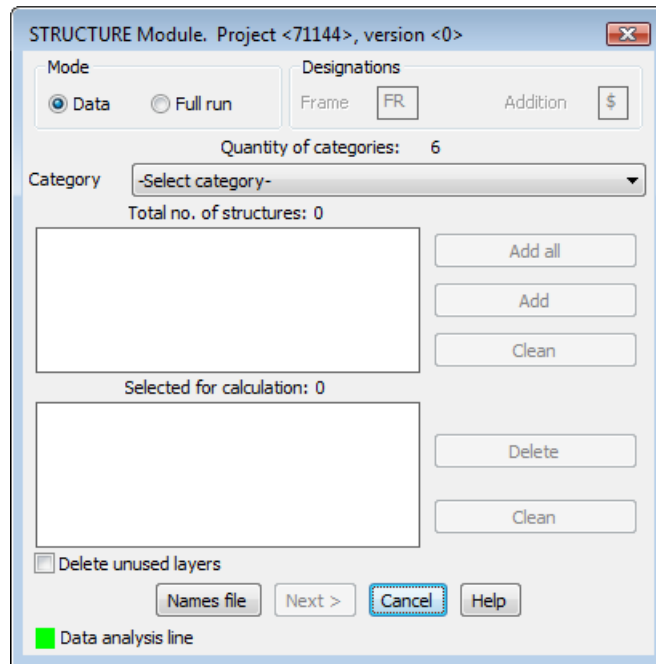StemName — stemframe (fore profile) layer name (e.g. _BT0_0_2_).

5.2.   Calculation modes

Command of structures creation can be run with menu item **STRUCTURE > Create structures** (see dr. 3) or with button of toolbar **Structure**, from dr. 4.

Command opens dialog box **STRUCTURE Module. Project <71144>, version <0>** (numbers of current project and data version are conditional) (dr. 7).

Drawing 7. Main window for building structures

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** there are two radio buttons:

– **Data** — work with data of selected structures (selective mode), with possible editing option;

– **Full run** — building all the structures described earlier in names file, without editing data.

Names file is *steel_<nn>.dat* (<nn> is a number of current data version), it is located in sub-folder *Steel* of the project folder (e.g. *D:\BSHIP\Projects\71144\Steel\steel_00.dat*).

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 first listbox displays all the structures of the selected category. The second one displays categories that were selected for session. In this document the first listbox (under text **Total no. of structures**) will be named *Category elements*, and the second listbox (under text **Selected for calculation**) will be named *Selected elements*. Work with categories and selection of structures are possible only in the mode **Data**.

If to set checkbox **Delete unused layers**, then unused 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 build-

ing structure lines on the opposite side then it is possible to mirror hull forms and to repeat building structures. To preserve line of the previous side clear checkbox **Delete structures**.

In the mode **Data** (selective) in the window **STRUCTURE Module. Project <71144>, version <0>** drop-down list **Category** (dr. 8) 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 categories.
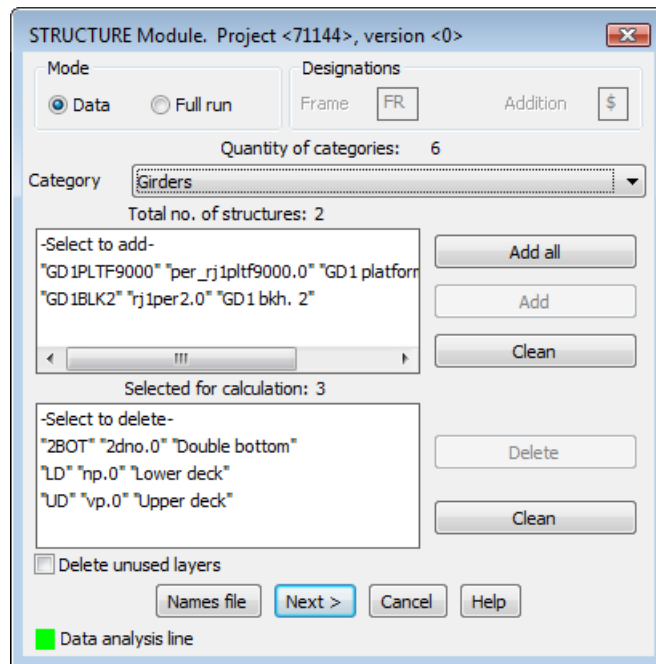
Drawing 8. Selecting structures for calculation

If click on category name (e.g. *Girders*), then all members of this category will be copied to the listbox *Category elements* (see dr. 8). In the line **Total no. of structures** quantity will be shown (on dr. 8 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 quantity of selected 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 (see dr. 8) 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. 9).

Drawing 9. 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.2" (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) is called.

On dr. 9 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**, **Bulk-heads**, **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 written inside names file (for **Full run** mode) or as they were selected in the dialog box on dr. 8 (for **Data** mode). If the correct sequence is violated then 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.

Checkbox **Delete unused layers** (see dr. 8) gives opportunity to purge drawing from extra layers.

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

Sign 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 for closing window or reading 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.

5.3.   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. *vp.0*) then after clicking button **Next** in the main window the dialog box **File <vp.0> missing** will be opened (dr. 10).
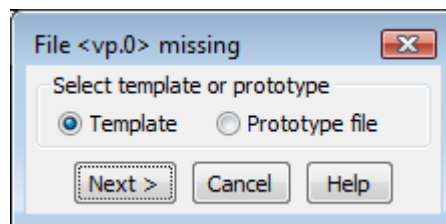

Drawing 10. Selection of template type

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 then dialog box **Select file as prototype <vp.0>** (dr. 11) and user must select the file from which data will be copied for created file vp.0 (to be edited later).
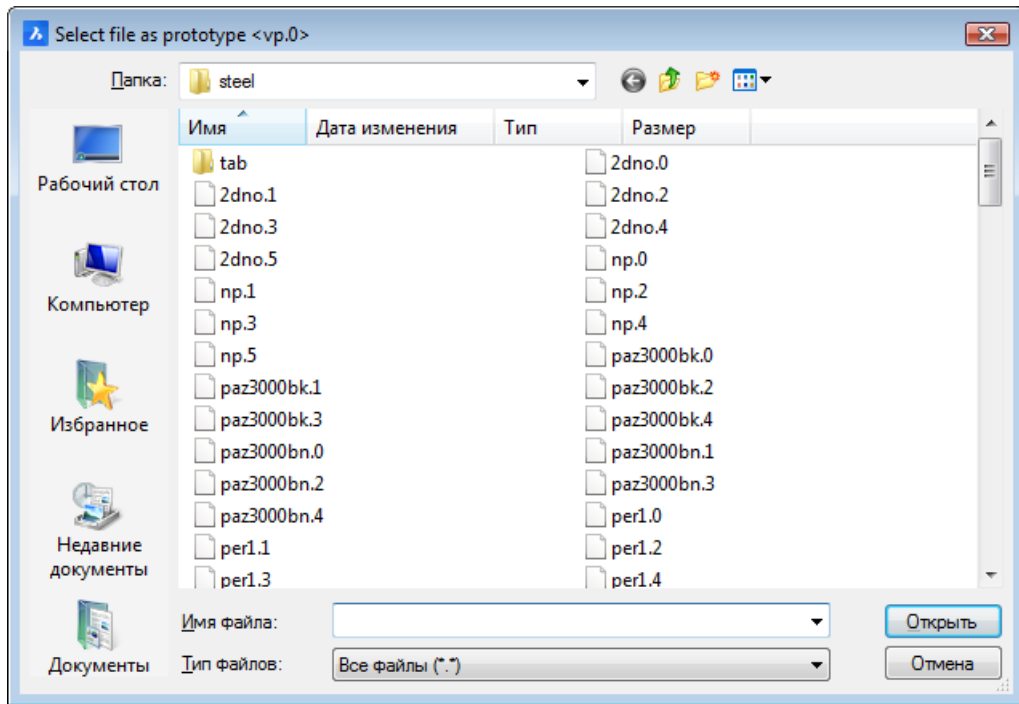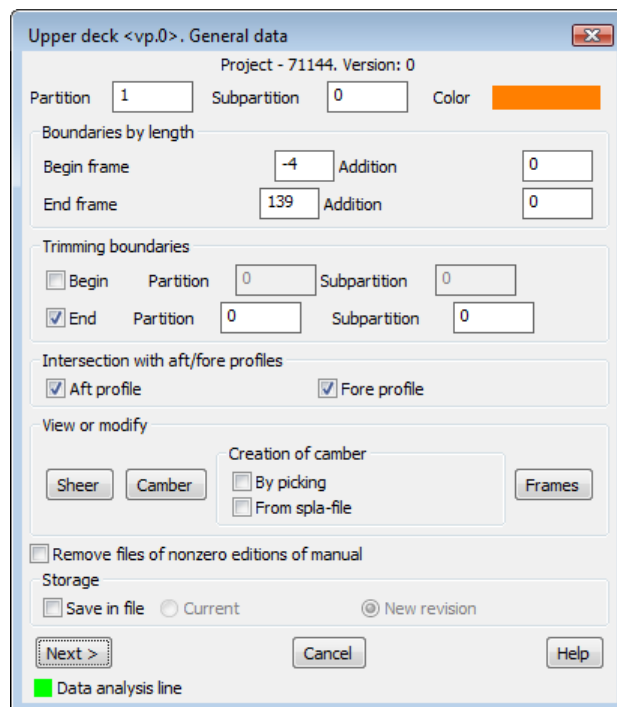
Drawing 11. Selection of file as prototype

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

5.4.   General data for structure

Next is dialog box **Upper deck <vp.0>. General data** (dr. 12), if upper deck is the first in the list of structures to be built (see dr. 8).

Drawing 12. Window **Upper deck. General data**

In the upper part of the dialog (see dr. 12) 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. 13).
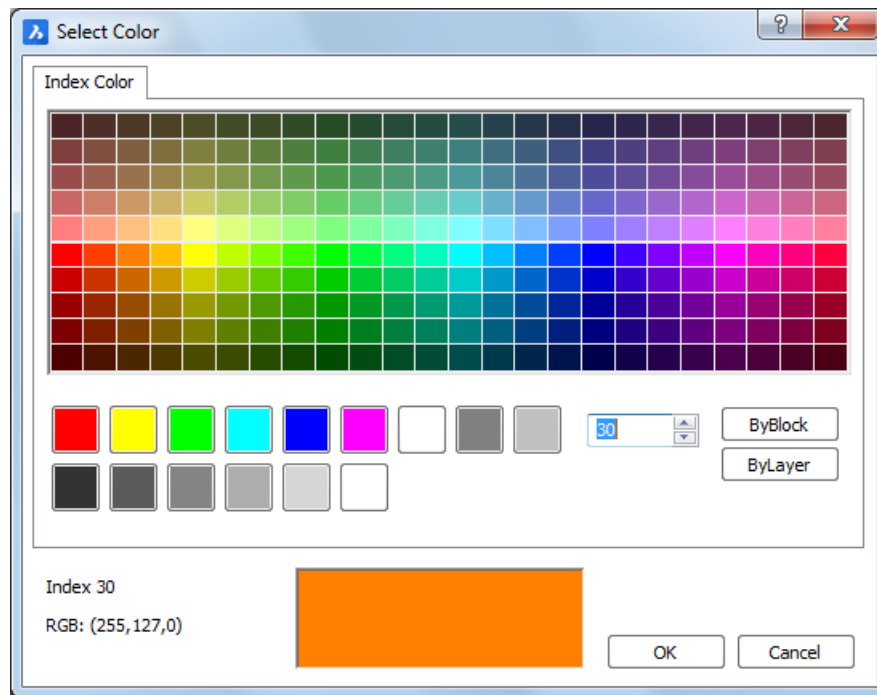
Drawing 13. Color selection dialog

Select required color (e.g. 30, kind of orange). After pressing **OK** dialog on dr. 12 returns and **Color** will get new color.

In the area **Boundaries by length** (see dr. 12) 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;

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

In the area **Trimming boundaries** (see dr. 12) 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);

14

– 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.

Note. If wireframe model is exported from big CAD system then it will a good step to request to export all the necessary auxiliary frame lines, because they will be needed for calculation of structures.

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).

5.5. Sheer

Button **Sheer** of the structure general data dialog (see dr. 12) opens window for editing sheer line data (dr. 14). 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.

Drawing 14. Sheer data

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 dr. 14 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 shown on dr. 14 there are three fields displaying sheer data for the segment selected in the listbox (no. 2):

– **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 (see dr. 14):

– **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**.

Data analysis line is used for messages on errors or warnings. Square sign 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 dr. 14 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 (see dr. 12).

5.6.   Camber

Click on button **Camber** in the dialog **Upper deck <vp.0>. General data** (dr. 12) opens dialog box **Upper deck <vp.0>. Camber** (dr. 15).

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.

Drawing 15. Camber data

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).

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.

Window on dr. 15 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;

– **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 BricsCAD).

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.

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 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.

Area **Zones** are 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**.

Buttons at the bottom of dr. 15:

– **Save**, closing window with saving data;

– **Cancel**, closing window with discarding changes in data;

– **Help**, viewing help topic for the window.

After **Save** or **Cancel** program returns to the previous window with general data (see dr. 12).

5.7.   Auxiliary frames

Dialog box **Upper deck <vp.0>. Auxiliary frames** (dr. 16) is being opened if in the dialog **Upper deck <vp.0>. General data** (see dr. 12) button **Frames** was clicked.

Drawing 16. 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 sign).

5.8.   Service window

In **Data** mode (selective calculation) after click on **Next >** button in the window **Upper deck <vp.0>..General data** (see dr. 12) there opens dialog box **Upper deck. Service** (dr. 17). 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 become disabled.
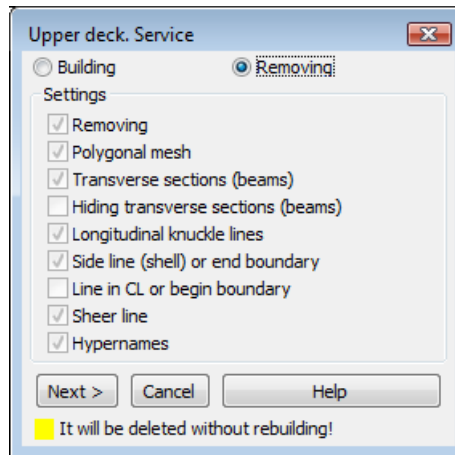
Drawing 17. Structure service data (mode **Removing**)

If radio button **Building** is activated then all the checkboxes in the **Settings** area are enabled (dr. 18):
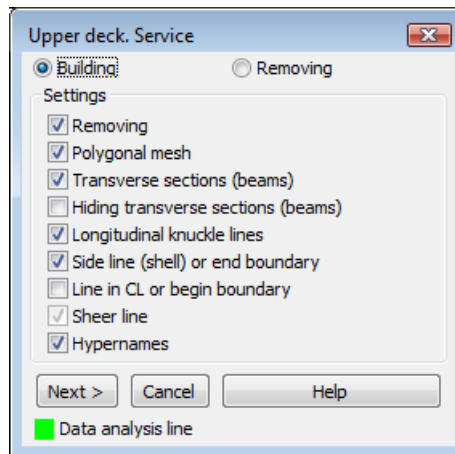

Drawing 18. Structure service data (mode **Building**)

– **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.

– **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. 19), usually with structure line layer name. Hypernames are shown on hovering cursor over the line, if hyperlinks are set active with command HYPERLINKOPTIONS.
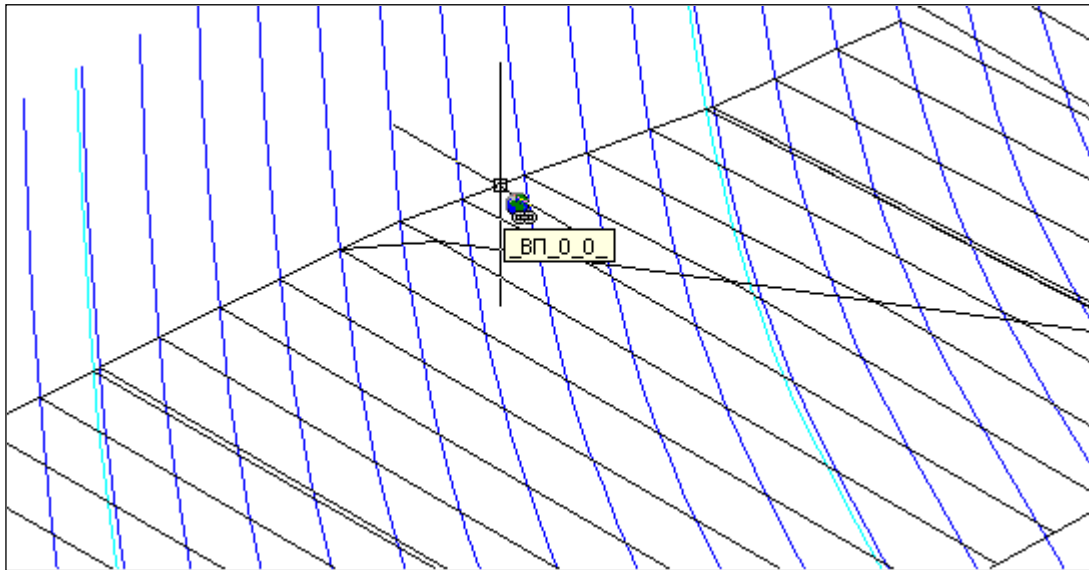
Drawing 19. Hypernames

Note. Hypernames are temporarily disabled.

If in the window **STRUCTURE Module. Project <71144>, version <0>** (see dr. 8) 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.

5.9.   Running calculation

Program begins structures calculation after click on button Next of dialog box **Upper deck. Service** (see dr. 18), if **Data** mode (see dr. 8) was activated.

If **Full run** mode was set then calculation begins immediately after click button **Next** in the main window (see dr. 7).
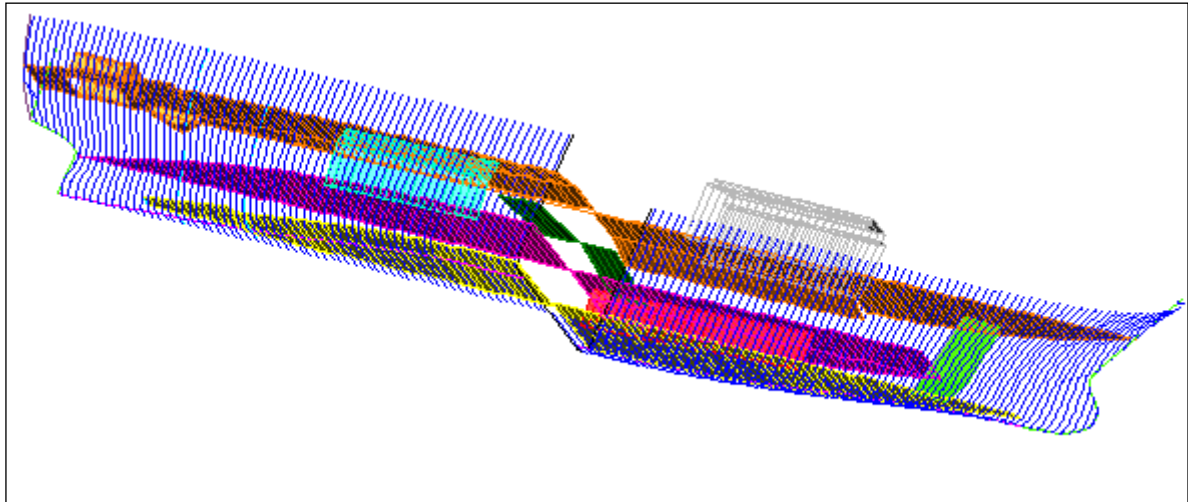
6. OUTPUT DATA

6.1. Module generates some kinds of graphical and textual data.

Each longitudinal structure building in wireframe model (see dr. 6) 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. 20 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 20. Built structures

6.2. Module **Structure** forms several text files. In subfolder *<PATH_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

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 contain module messages. New lines are appended to the file but if file size exceeds 60 Kb, then it will be cleaned.

6.3. Sample results of building structures are given in dr. 21 and 22. Visual control of results is usually run with command 3DORBIT.
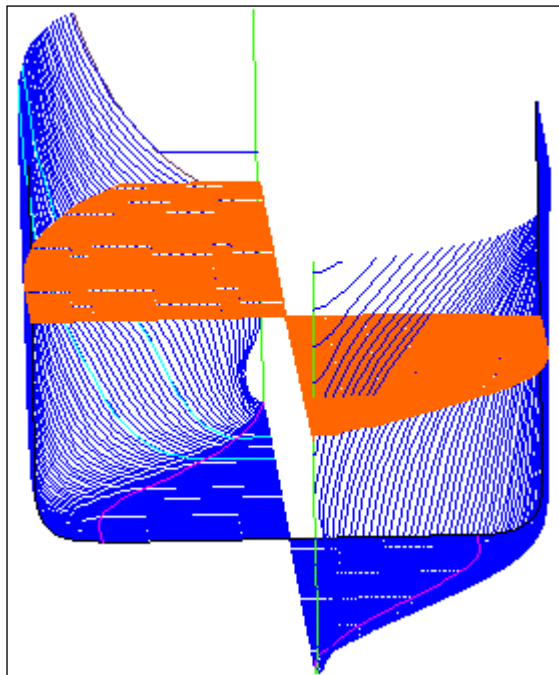
Drawing 21. Sample building only one structure

On dr. 21 there is a deck built in mode **Data**, and on dr. 22 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.

7. INPUT ERRORS DESCRIPTION

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

After correction of data calculation may be continued.
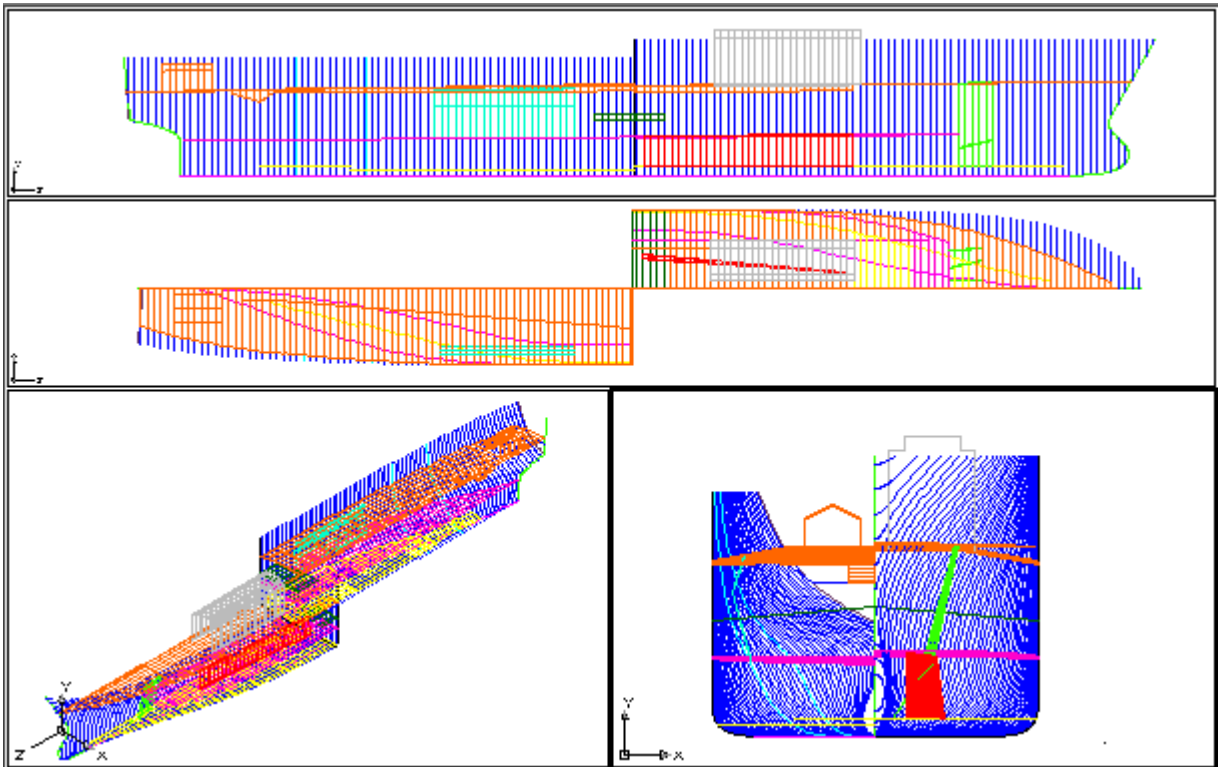
Drawing 22. Sample results of mode **Full run**

Here are some error messages supported with red square sign 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 graeter 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 is 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;

– "Height should not be greater than 39000" — height cannot be greater than double maximum height ("H" -19,5) for this project.

Messages with yellow square sign does not prevent work continuation. User may edit some data or go on working. Here are some yellow signed 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 according to data from spacing table.

8. CALCULATION OF SHELL EXPANSION LINES

8.1. 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 the good height and not intersected with real model objects) and can be exported as very important lines for creating a difficult document that is shell expansion drawing.

8.2. Before calling expansion programs it is necessary to open wireframe model with B-Ship organization in the graphical editor (BricsCAD). Then user must set the required project name (number) in the dialog box **Managing projects and data versions** opened with menu item **STRUCTURE > Manage data versions**. More on projects and data versions see hereinafter.

Expansion calculation is launched with menu item **STRUCTURE > Shell expansion** (see dr. 3) or with button [icon] of toolbar **Structure** (see dr. 4). Command opens dialog box **Expansion initialization for pr.NNNNN** (NNNNN is current project name) (dr. 23).
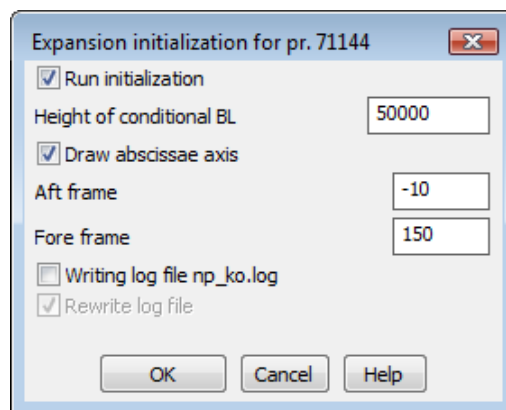
Drawing 23. Window **Expansion initialization for pr.**

After running initialization user will get request for selecting lines that are to be expanded. After selection press **Enter** and these lines will be generated.

8.3. Dialog box **Expansion initialization** (see dr. 23) allows user to enter initialization parameters. For running initialization procedure it is necessary to set checkbox **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.

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

Setting checkbox **Run initialization** gives opportunity to define 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).

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). They are to be fitted with spline tools. Color and layer of expanded lines are inherited from their parent objects (with one modification: layer name of the expanded line has subscription symbol "_" as prefix).

Dr. 24 shows sample result of calculating expanded side lines for decks, platforms, seams, etc., drawn over the source 3D model.
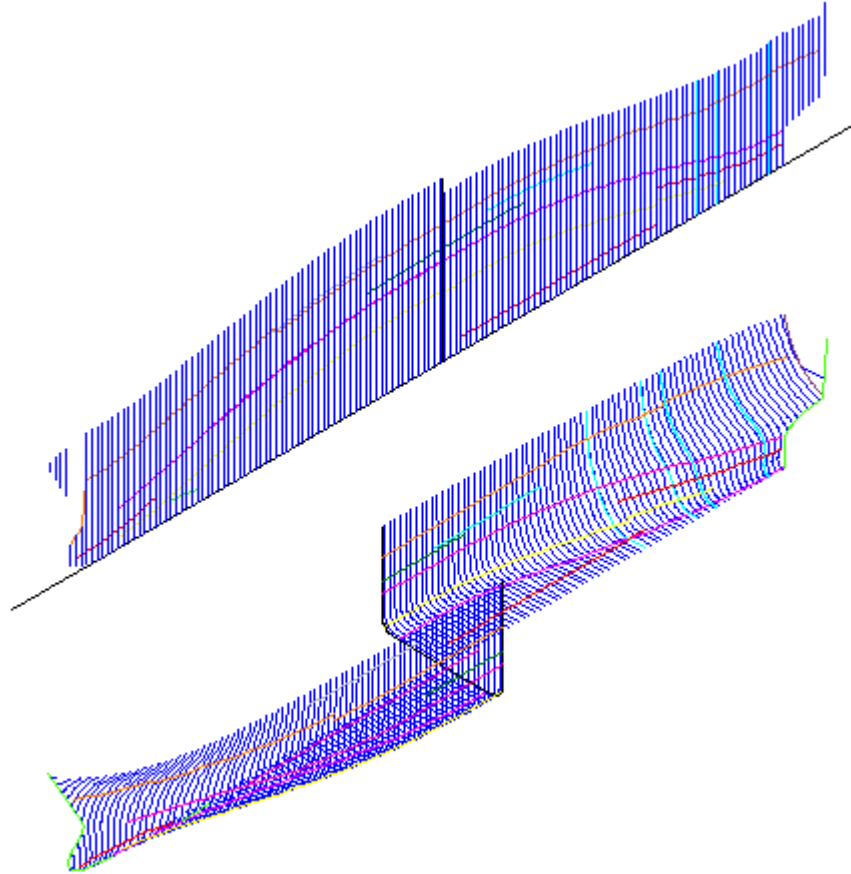
Drawing 24. Sample expanded lines

For generation of the drawing document it is necessary to export built lines (conditional BL, expanded lines) to a separate file — for example, with command WBLOCK.

Final drawing operations (copying aft/fore profile lines, creating transverse stringers, inscription insertion, etc.) are done with usual tools of graphical processor.

9. PROJECTS AND DATA VERSIONS MANAGEMENT

9.1. Current values of project, data version can be viewed with menu item **STRUCTURE > Manage data versions** (see dr. 3) or with button of toolbar **Structure** (see dr. 4). This opens dialog box **Managing projects and data versions** (dr. 25), which displays project name and data version number.
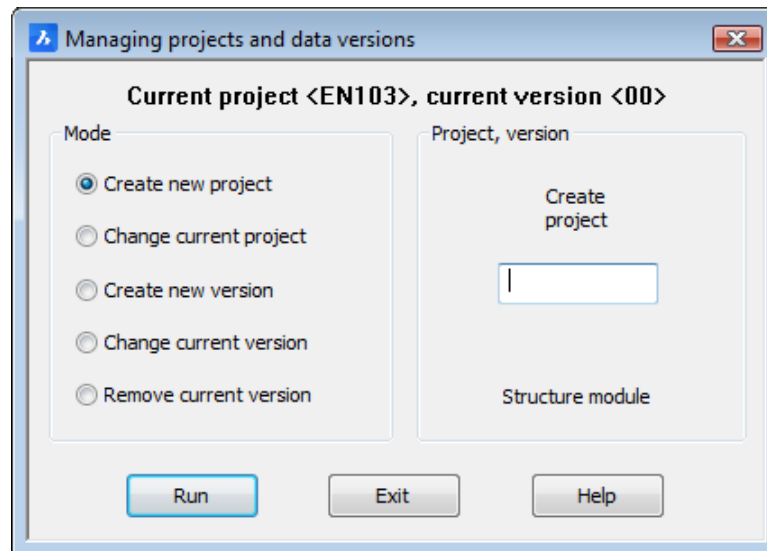
Drawing 25. Dialog box **Managing projects and data versions**

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**.

9.2. 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** (see dr. 25). Other command with the same functions is menu item **MODEL > New project**.

To create new project in the dialog box **Managing projects and data versions** (see dr. 25) it is necessary:

- 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 info about new project.
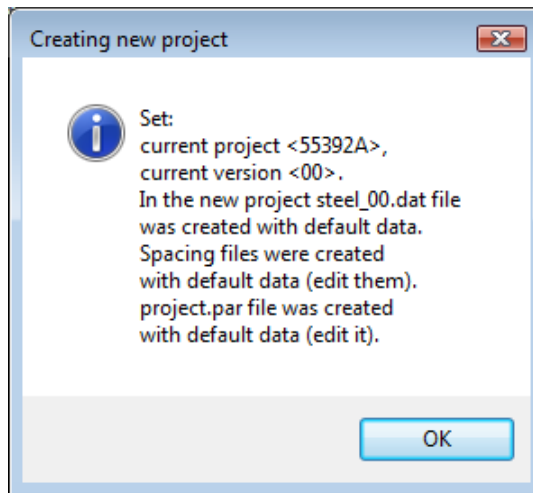
Drawing 26. Message about successful creation of new project folder

After creation of new project folder it is obligatory to edit files of general parameters and spacing tables, as they are filled by standard initial data.
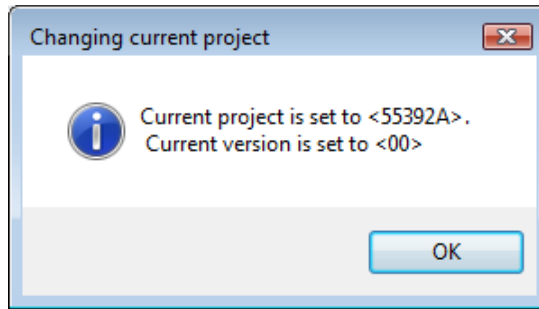
9.3. To change project in the dialog box **Managing projects and data versions** (see dr. 24) it is necessary:

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

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

- to click on button **Run**.
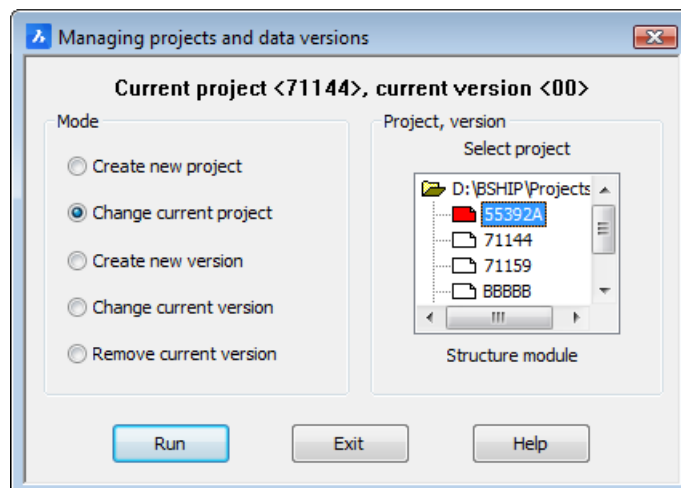
Drawing 27. Changing current project

After clicking button **Run** there appears window 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

9.4. 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**.
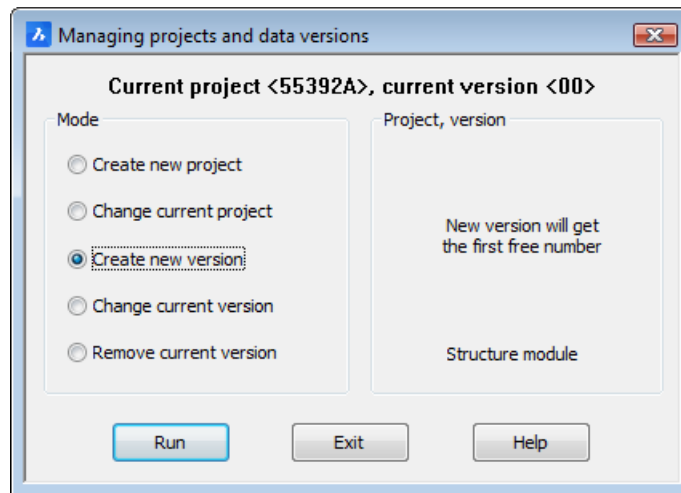
Drawing 29. Creating new data version

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.
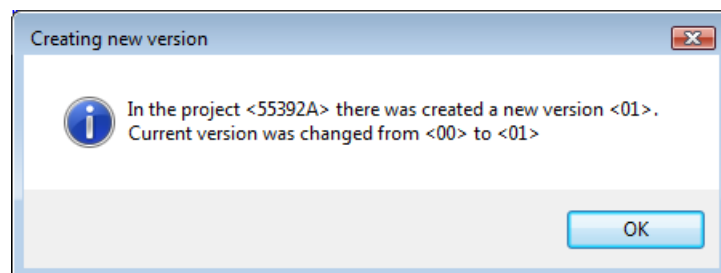
Drawing 30. Message about creation of new data version

9.5. 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 list **Select version** with numbers of existing versions for the current project to select number of the version that must become current (dr. 31);

- to click button **Run**.
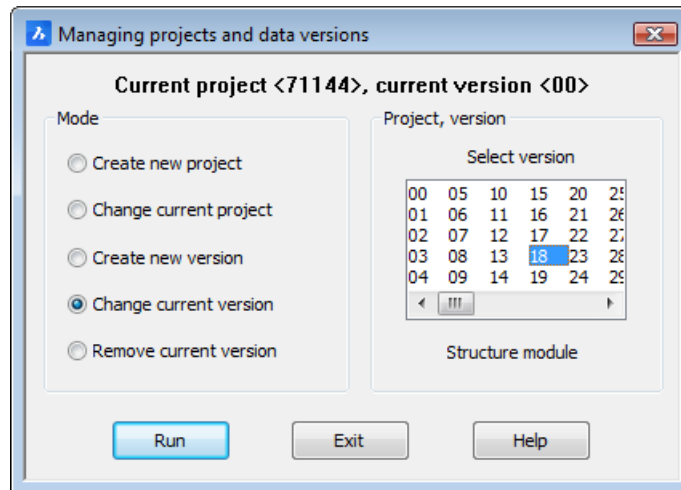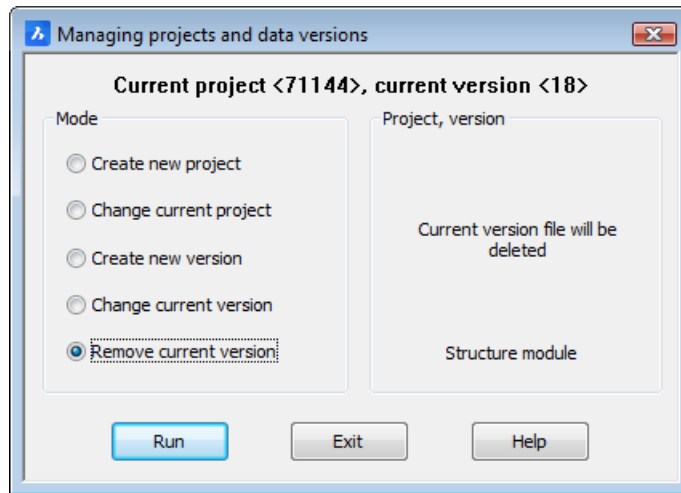

Drawing 31. Changing current data version

After selecting version number and clicking **Run** there appears a message (dr. 32).
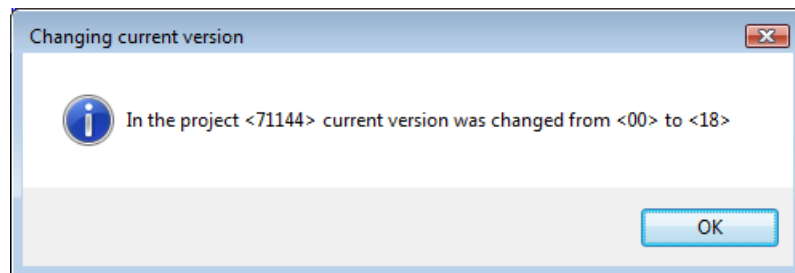
Drawing 32. Message about changing current version

9.6. Any data version file can be deleted, but it must be current. If necessary to delete it then do the following:

- set this data version file as current (see dr. 31);

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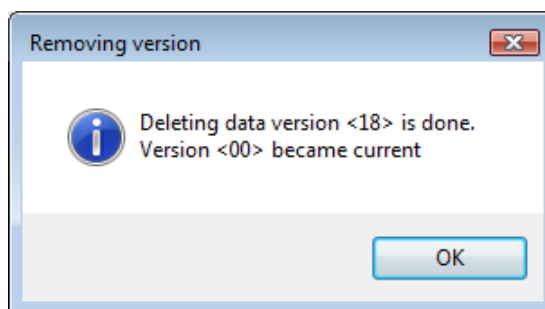
- click button **Run**.

Drawing 33. Deleting current data version

Deleting is dangerous operation, that's why program warns (dr. 34).
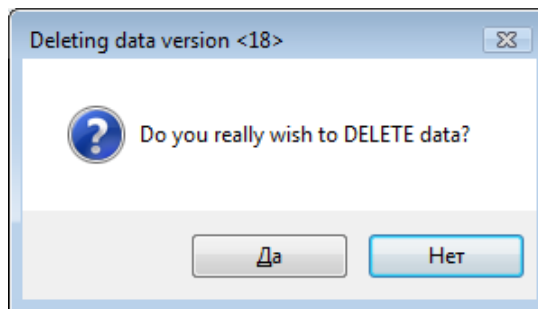
Drawing 34. Warning before deleting data version file

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).

Note. Basic version cannot be deleted.

10. LOFTBOOK CALCULATIONS

10.1. Loftbook is a document containing table of coordinates for ship shell geometric model lines, with predefined increments. It can be divided into two parts. Part 1 includes coordinates of frame lines by waterlines and by buttocks. Part 2 includes coordinates of structure side lines by frames (for decks, platforms, stiffeners, etc.).

Model frame lines (entities) involved in calculation must be of type 2D-POLYLINE, 3D-POLYLINE or LINE. All the lines must be located on layers with names satisfying geometric model organization, used in B-Ship:

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

Here are name components:

<_> – subscription symbol ("_");

<type> – two symbol name (FR, WL, BT, UD, Bkh, etc.);

<number> – number (integer) of the main line;

<addition> – empty string for zero addition (in case of a main frame) or string of type "$" + <positive integer> (in case of an intermediate frame moved from the main frame to fore);

<partition> and <subpartition> – integers from –32768 up to 32767.

Sample layer name: _FR25_0_0_, _FR25$250_0_0_ or _UD12_2_0.

For working with loftbook there are **STRUCTURE** menu items:

- **Loftbook, p.1**, runs calculation of part 1;

- **Loftbook, p.2**, runs calculation of part 2;

- **View loftbook**, viewing of earlier created loftbook file *.plk.

Both types of calculation start with model analysis. If current drawing is not a model file or there are no lines on frame layers then an error message will be displayed. If model is found then there is a message with total quantity of found main and intermediate frame lines (dr. 36).
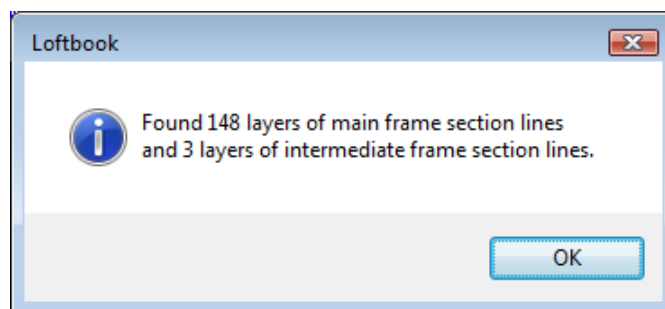
Drawing 36. Message after finding model frame lines

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

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**.
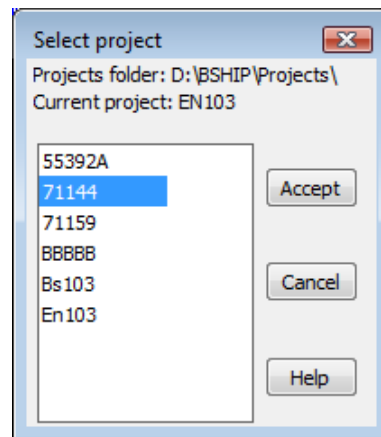
Drawing 37. Window **Select project**

In case of click on **Cancel** user refuses from calculation.

10.2. For calculation of loftbook part 1 there are used menu item **STRUCTURE > Loftbook, p.1** or button ⬚ of toolbar **Structure**.

Calculation starts with model layers analysis. If model is good (see dr. 36) then dialog box **Select project** opens (see dr. 37).

After project selection program makes an input request in command line: **User name**:. User must enter any user name (to be added to the document). Then program asks to choose calculation type, in dialog box **Loftbook calculation. Project NNNNN** (dr. 38).
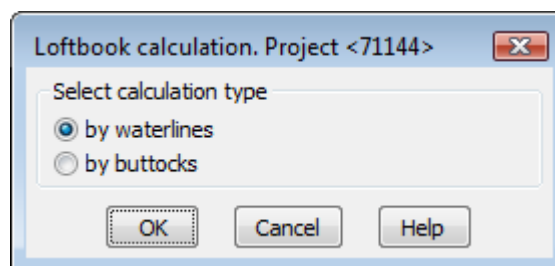
Drawing 38. Window **Loftbook calculation** (mode **by waterlines**)

The default value in the radio column **Select calculation type** is **by waterlines**. If it satisfies then click **OK**. After finishing calculation by waterlines the window will be opened once more and in the radio column **Select calculation type** there will be set (suggested) radio button **by buttocks** (dr. 39).

If this is OK then click button **OK**, otherwise set type **by waterlines** again or click **Cancel** button. Calculation types can be selected in any sequence. Button **Help** opens help topic.
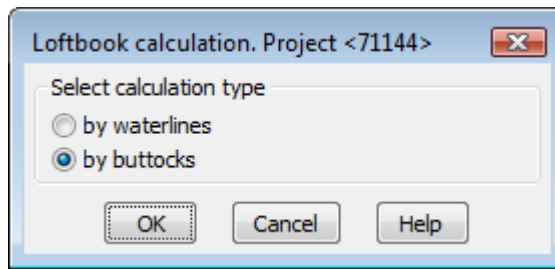
Drawing 39. Window **Loftbook calculation** (mode **by buttocks**)

Next step asks to set document parameters in the dialog box **Document settings** (dr. 40).
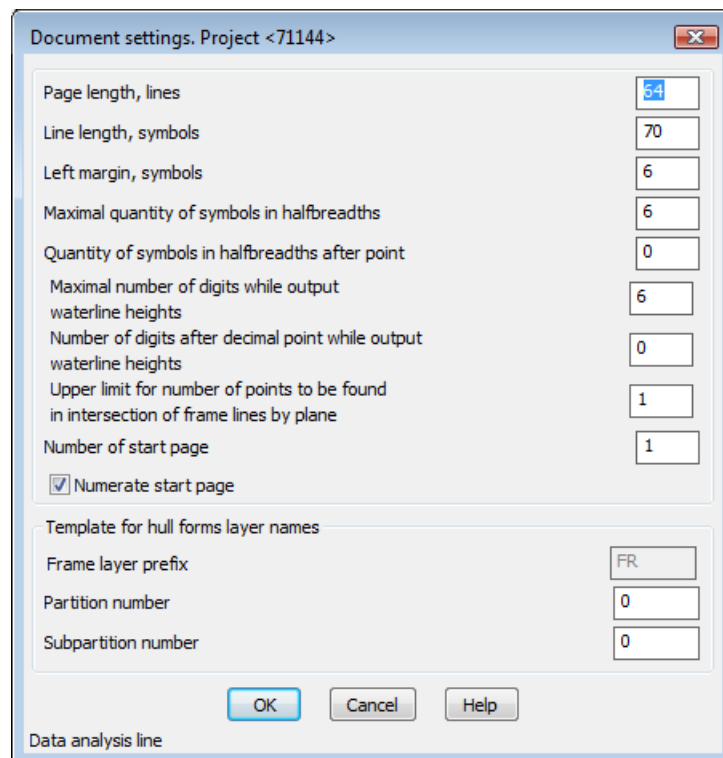
Drawing 40. Window **Document settings**

It is necessary to define the following documents parameters for the future plain text file:

- page length (allowed 9 - 64);

- line length (allowed 64 - 128);

- left margin (allowed 1 - 10);

- maximal number of digits in the halfbreadth (height) values (allowed 6 - 9);

- number of digits after decimal point in halfbreadth (height) values (allowed 1 - 3);

- upper limit of points to be searched in the operation of intersection line by the frame plane (allowed 1 - 9);

- number of start page number (allowed 1 or more);

- partition number for shell frame lines (allowed from -32767 to 32768);

- subpartition number for shell frame lines (allowed from -32767 to 32768).

All these parameters have default values that can be edited. If user enters unallowed value or empty value the in the **Data analysis line** there will be an error message, for example: "*Inadmissible value!*". And cursor will be moved to the field with an error.

If calculation is run for the first time in the current session then user must select state of the checkbox **Numerate start page** (by default it is set on). It is the second time (or more) then start page number is automatically generated, its value is set into the window, and checkbox **Numerate start page** will be disabled.

After clicking **OK** user is directed to the next step, that is defining limits for which calculation is intended to be done. Window **Project frame boundaries** (dr. 41) opens. It shows minimal and maximal possible frame numbers.
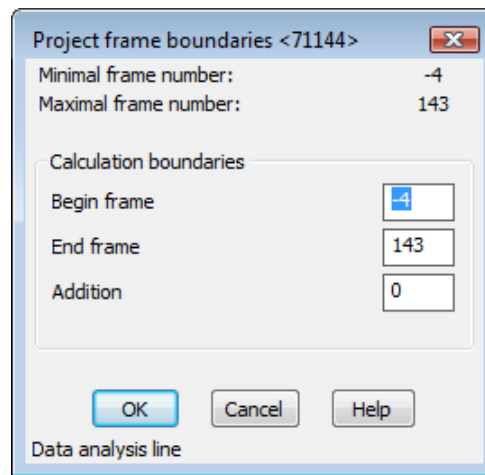

Drawing 41. Window **Project frame boundaries**

In the area **Calculation boundaries** user can change initial values of begin frame (aft), end frame (fore) and addition. If **Addition** is not zero then calculation will handle only intermediate frames with this addition value, no other frames will be used.

Begin and end boundaries must not be outside limits written to project spacing tables. Otherwise error message in **Data analysis line** is generated, e.g.: "*Frame number is out of spacing table!*", " *Frame abscissa is out of spacing table!*" or "*Addition of frame is > or = spacing!*".

Corresponding field will be highlighted. After correction click **OK**.

OK leads to defining waterline heights (or buttock halfbreadths if it is calculation by buttocks). Heights can be divided into groups with constant increment. Module suggests a list of default heights, with different increments in several groups. This default list is shown in the window **Do you agree?** (dr. 42).
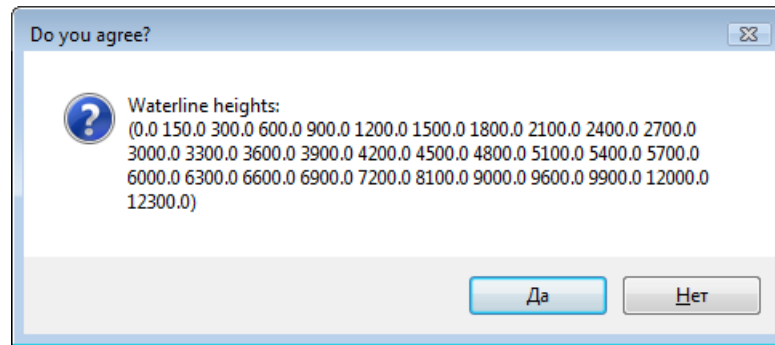
Drawing 42. Window **Do you agree?**

If you agree then click **Yes** (Да). If you wish to enter other heights then click **No** (Нет) and dialog box **Waterline heights** (dr. 43) will be opened, for the first group of heights:
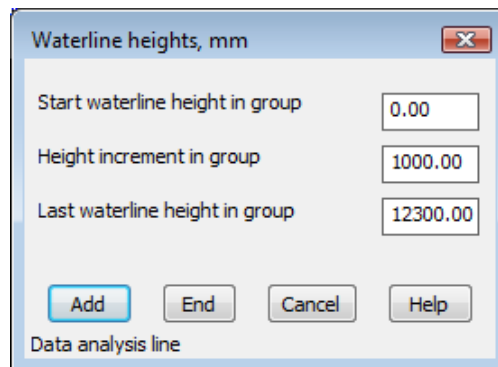
Drawing 43. Window **Waterline heights**

Here are the following parameters:

- **Start waterline height in group**;

- **Height increment in group**;

- **Last waterline height in group**.

For the first group default start height is 0. User can enter his own values. After clicking button **Add** the first group data will be saved and the window **Waterline heights** (see dr. 43) will show some values for the next group. Value in the field **Start waterline height in group** for the next group will be taken from the last waterline height of the previous group. Moreover field **Start waterline height in group** will be disabled. User must enter data for the next group and click **Add** or click **End** that means finishing groups (no new groups needed).

Program checks the last waterline height value. If it is entered inexactly then it will be modified a little and new value will be displayed in the field **Last waterline height in group**.

After finishing heights definition and clicking button **End** program will display new list of heights in the window **Do you agree?**. User must click **Yes** (Да) when heights list is finished. This launches calculation itself, for loftbook, part 1.

In a similar way there can be entered buttocks data to create loftbook table by halfbreadths. It is done in dialog box **Buttock halfbreadths** (dr. 44).
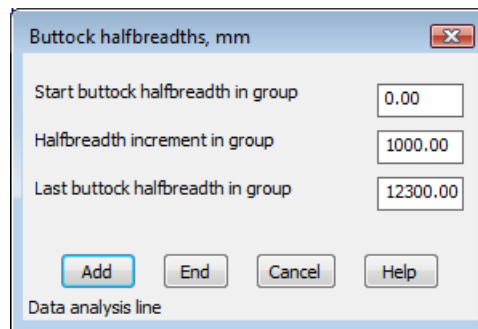

Drawing 44. Window **Buttock halfbreadths**

After end of calculation program will suggest to enter name of the resulting document file with .plk extension.

10.3. For calculation of loftbook part 2 there are command **STRUCTURE > Loftbook, p.2** and button [2] of toolbar **Structure**.

Calculation starts with model layers analysis. If model is good (see dr. 36) then dialog box **Select project** opens (see dr. 37). After project selection program requests in command line: *User name:*. User must enter any user name (to be added to the document).

Next is dialog box **Document settings** (dr. 45).
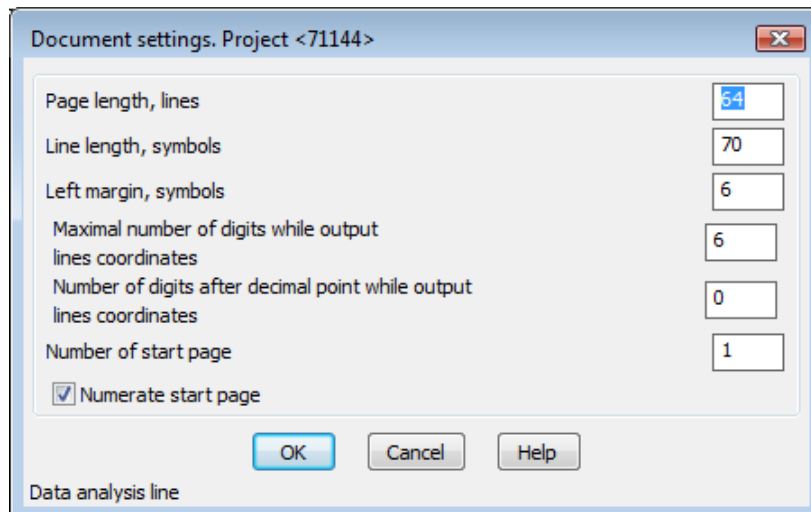
Drawing 45. Window **Document settings**

Here are the following document parameters:
- page length (allowed 11 - 64);
- line length (allowed 64 - 128);
- left margin (allowed 1 - 10);

- maximal number of digits in the halfbreadth (height) values (allowed 6 - 9);

- number of digits after decimal point in halfbreadth (height) values (allowed 1 - 3);

- upper limit of points to be searched in the operation of intersection line by the frame plane (allowed 1 - 9);

- number of start page number (allowed 1 or more).

Calculation limits are defined in the next window **Project frame boundaries** (dr. 46).
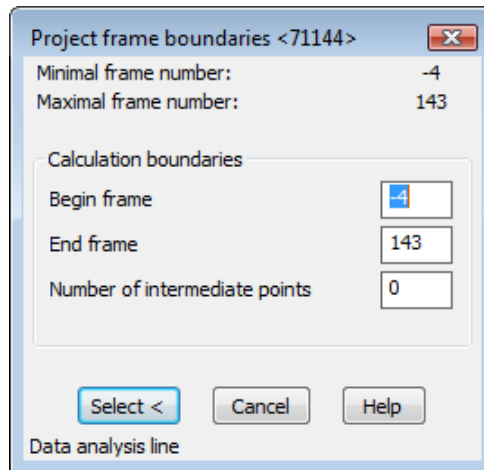
Drawing 46. Window **Project frame boundaries**

In the area **Calculation boundaries** user can change initial values of begin frame (aft), end frame (fore) and quantity of intermediate points between main frames (by default it is 0). Calculation will be done for main frames and for intermediate frames (if they are requested by **Number of intermediate points**).

Begin and end boundaries must not be outside limits written to project spacing tables. Otherwise error message in **Data analysis line** is generated, e.g.: "*Frame number is out of spacing table!*", "*Frame abscissa is out of spacing table!*".

After defining calculation boundaries it is necessary to pick the longitudinal lines to be involved into the process. User must click button **Select <**. Dialog box will be closed and program will request: "*Pick line: Select objects:*". End of selection is pressing **Enter**. The window **Project frame boundaries** (see dr. 46) opens again. On **Select <** user can select some more objects. If all the objects are already selected then user must click button **Cancel**. If there is a set of selected lines then calculation will be launched (if selection is empty then session will be ended).

After calculation program will suggest to enter name of the resulting document file with .plk extension.

10.4. Menu item **STRUCTURE > View loftbook** and button 🔍 run command for viewing loftbook file with .plk extension. Dialog box will be opened to select PLK file. After selecting file and pressing **Open** Notepad will open the file.