

INTRODUCTION

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I. GRAPHIC EDITOR

INTRODUCTION

In this chapter the graphic editor and the organization of dialogue with system, start and an output from system, work with the system menu are described.



INPUT DEVICES

A mouse and a keyboard are used as the input devices for the SEA SOLUTION system.

MOUSE

The mouse is the principal means of the user and system interface except for the text entry.

The system allows using a mouse with two or three buttons. The buttons are defined in the following way:

for a three- button mouse:

- left-hand button - "**Space**" (corresponds to "**Space**" (blank) keyboard button);
- mid button - "**Enter**" (corresponds to "**Enter**" keyboard button);
- right-hand button - "**Esc**" (corresponds to "**Esc**" keyboard button);

for a two- button mouse:

- left-hand button - "**Space**";
- right-hand button - "**Esc**".

To use a three-button mouse you will need a special drive installed which will support operations in the three-button mode.

KEYBOARD

The keyboard is used for entry of text or numerical expressions and for quick entry of commands.

PRESSING MOUSE BUTTONS AND KEYBOARD BUTTONS

Pressing a button or a key means that such button or key is pressed and immediately released. In various publications you can meet the term "tap" or "click" for the notion.

Symbol "+" between the letters or conjoint letters are used to specify that the keys must be pressed simultaneously. For example, when you are required to press "**Alt+W**" or "**AltW**":

- press the "**Alt**" key and hold it depressed;
- press the "**W**" key;
- release both keys.

KEYS "SPACE", "ESC", "ENTER"

The mouse buttons correspond to the following keyboard keys: "**Space**", "**Enter**" and "**Esc**". In the manual you will meet the following instructions: "press "**Enter**", "**Space**" or "**Esc**", which you can do with the use of the mouse or keyboard.

"**Space**" is used to select a menu item, highlight screen elements, enter points and etc. It corresponds to the left button of the mouse.

"**Esc**" is used to cancel a command execution, exit the dialogue boxes and etc. It corresponds to the right button of the mouse.

I. GRAFIC EDITOR

"Enter" is used to complete the command entry or confirm the prompt. It corresponds to the mid button of the mouse.

OPEN AND CLOSE SYSTEM

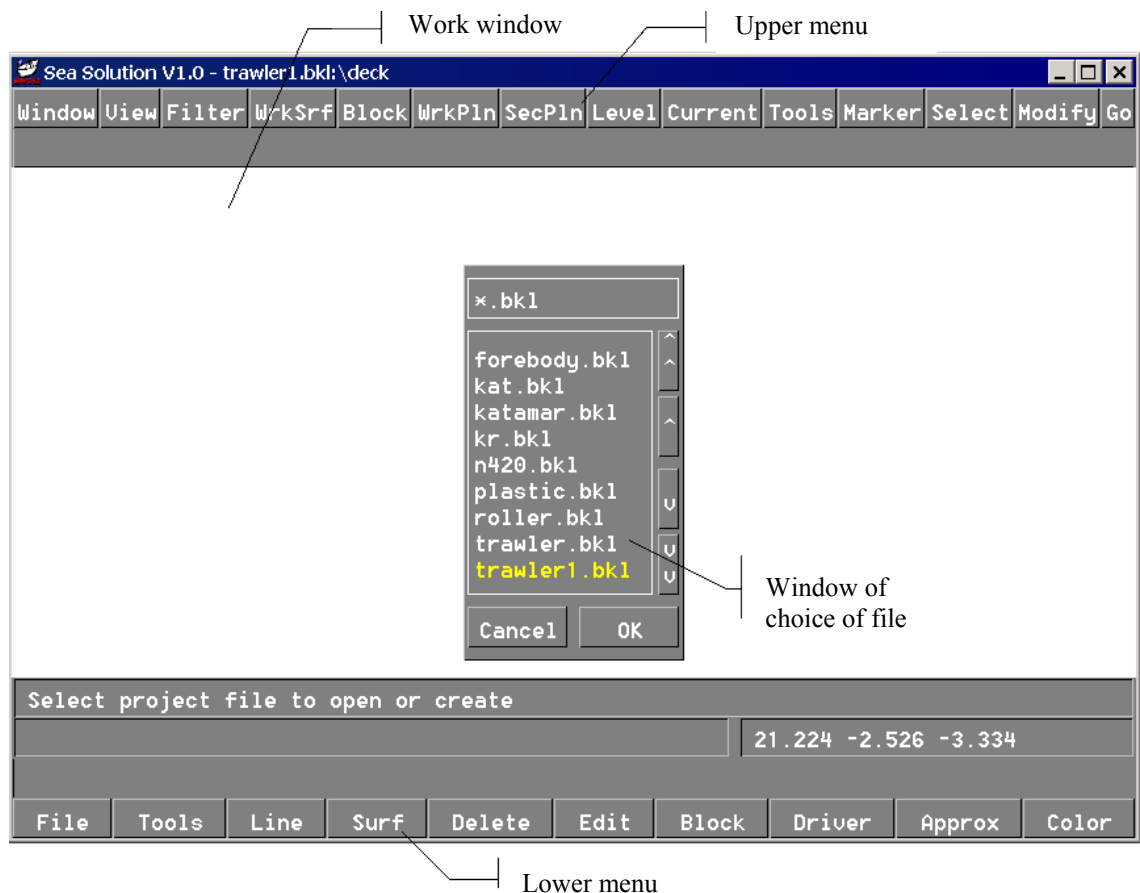
For start of work in Sea Solution you should press on an icon corresponding to the system Sea Solution

For output from the system you should press

"Alt+X" ore "File/Exit" (bottom menu)

GRAPHICS WINDOW

The central part of the screen contains the working area of the editor. An image is displayed and a mathematical model is formed (input and editing of lines, surfaces and etc.) in the working area.



SYSTEM MENU

The system menu represents of two lines of buttons in top and below of the screen. The majority of items of the menu contain a submenu which items are carried out commands.

The system menu consists of menu of project commands (lower menu) and menu of service commands (upper menu).

The lower menu consists of commands of creation of mathematical model of the ship (impute, edit, delete), commands of exchange of dates (export in DXF, import from DXF) etc.

The upper menu consists of commands of input to the screen (current view, current window), visibility of elements, object snap and some others.



CURSOR

The form of a marker depends on in what zone of the screen it is.

The form of cursor. Description.



The cursor is within the menu or dialogue box field



The cursor is within the software window



The object snap is active



Special mode of the marker



This form is assumed when a text is entered in the prompt or in the dialogue box



Input of text ore numbers in the line of output

Buttons

The basic means of a choice of items of the menu and input of commands are buttons of the menu.

The choice is carried out by pressing of the corresponding button. To press the button, it is necessary to deduce on it a marker and to press "Space" (on the keyboard or the left button of the mouse).

If the item of the menu or a command is inaccessible, you will hear a sound signal.



Button is not pressed

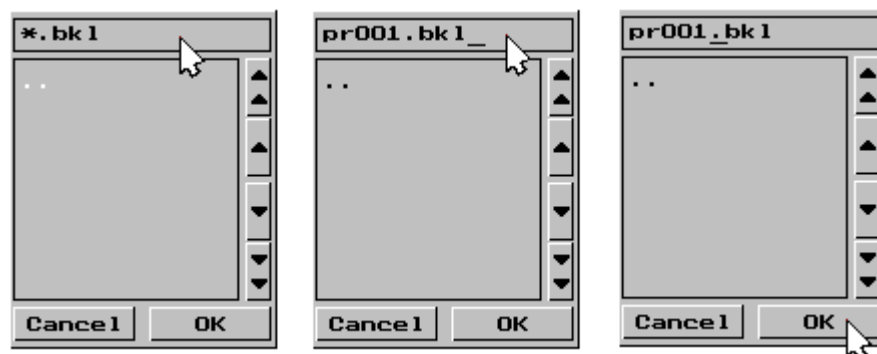
Button is Pressed

Loading

If you have started work in the system for the first time (any file of the project is not created) or have started system without automatic loading the project (i.e. the file of the project is not open), in the center of the screen you see a dialogue window of a choice of files of the project. You should create the new project or to load any of existing.

If you want to create a new project, you should move a marker to a line of a mask of search, as it is shown at the picture and press "Space" (the left button of the mouse). (Instead of it, it is possible to press a key "Tab" on the keyboard.) A text cursor "_" will appear In the line.

You should enter a name of a file of the project, using keys "← →", "Del", "BackSpace" and keyboard. Then press "Enter" or move a marker on button "OK" and press "Space".

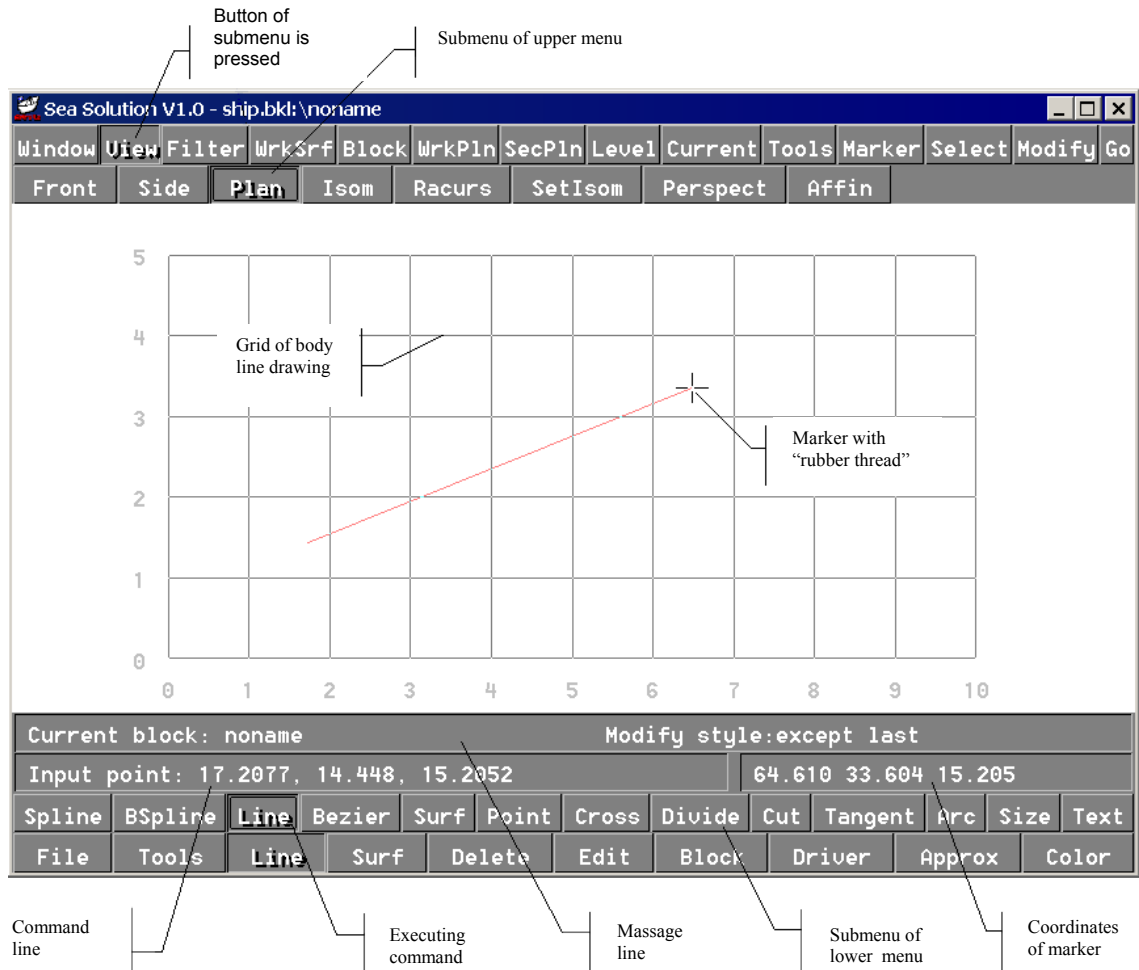


I. GRAFIC EDITOR

After that you should press **“Enter”** ore button **“OK”** on the question: **“Will create a new project?”**

A body line grid will be appeared in the work window after you have loaded of the project

The grid has sizes 100 x 50 x 50 meters if the project is new (in chapter IV “Project procedures” you will study to change the body line grid).



COMMAND LINE

The line is used for input text or numerical data and editing of the element coordinates (points, lines and etc.).

For example, when entering a point in the prompt **“Input point:”**, the system suggests several input coordinates. If you press **“Enter”**, you will introduce a point having such coordinates.

With the help of the following keys: **“←→”**, **“Del”**, **“BackSpace”** you can edit the suggested coordinates. Press **“Enter”** to complete the entry.

If the required coordinates considerably differ from the input coordinates, enter such coordinates right away. The newly entered data will replace the input coordinates suggested by the system.

Enter **“Esc”** to cancel the keyboard entries.



CHOICE OF MENU ITEMS

When you press button **“View”** in upper menu it will appear a submenu. If the button **“View”** is pressed it means that the submenu is active. If the button **“Side”** is pressed, it means that a current view is **“Side”**.

Further for a designation of the button or item of a submenu will be used "press View/Side in upper menu".

INPUT OF COMMANDS

If you press **“Line/Line”** in lower menu it will be corresponded to a command **“input line”**. You will see inquiry of the system **“Input point: 0, 0, 0”** – you should input a beginning point of a line. You should input point or cancel this command.

If you want to cancel of this command you should press **“Esc”**.

INPUT OF POINT. INPUT OF LINE.

Move a marker to a working window. Enter the first point of a line, for this purpose press "Space". If you move a marker you will see a **“rubber thread”**. You should enter the second point. The entered line and two points are appeared on the screen. If you move marker you will see a **“rubber thread”** again. You can continue an input of line. If you want to cancel input you should press **“Esc”**.

GENERAL DIALOGUE STRUCTURE

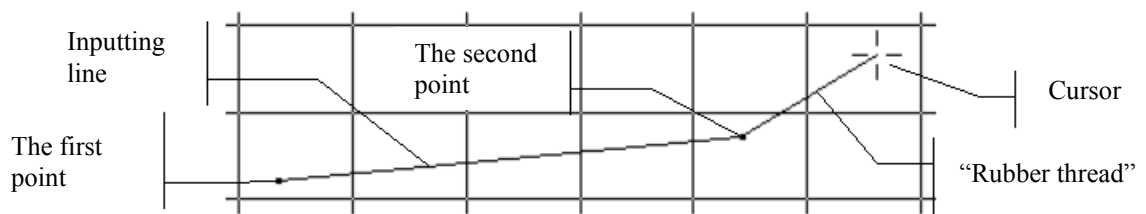
A dialogue between the user and the system is carried out in the following manner: with the help of system menu the user enters a command. Executing the command the system displays prompts. The user then executes the prompt or aborts the command. If execution of the command is impossible due to any reason, the system will inform the user on this and suggest repeating or aborting the operation.

EDITING LINES

Text lines or numerical expressions in various dialogues are edited with the help of the following buttons: **“← ↑ ↓ →”**, **“Del”** and **“BackSpace”**.

To complete the input use the **“Enter”** key and to cancel the entry press **“Esc”**.

If you start editing by first pressing any alphanumeric key, the system will assume this action as an entry of a new line and delete the initial text in the line. If you press **“Esc”**, the deleted text will be restored.



HOT KEYS

Frequently used commands can be entered with the help of "hot keys". For example **“Alt+X”**- close system.

The list of "hot keys" is resulted in Appendix "Hot keys".

REQUEST FOR CONFIRMATION

In some cases the system will prompt you to confirm the command execution. By pressing the "**Space**" or "**y**" keys you agree to the command execution and by pressing the "**Esc**" or "**n**" keys you abort. "**Enter**" is the default setting (as specified in the brackets in the end of the question).

INTERRUPTION OF CONTINUOUS OPERATIONS

Any continuous operation such as the display operation, export to DXF-file or correction can be interrupted by pressing the "**Esc**" key.

If execution of the operation exceeded a certain period, the system will request for a confirmation:

"Do you really want to stop the process (y)? "

The general organization of dialogue

Dialogue of the user with the system looks as follows. The user by means of the system menu enters commands. Executing commands, the system gives out inquiries. The user carries out inquiries, or cancels a command. If performance of a command for any reasons is impossible, the system informs on it or suggests to repeat performance of a command, or cancels its performance.

CALCULATOR

A line calculator is implemented in the *SEA SOLUTION* system. The calculator functions are available as during subtractions in the calculator dialogue box (recalled by pressing "**AltC**").



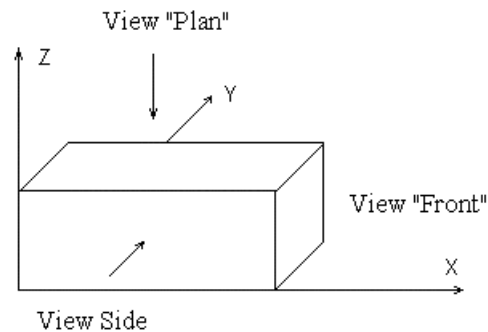
The expressions can also be used in the prompt when entering the numerical expressions (for example, when entering a point coordinates). The calculator possesses a number of mathematical functions and a number of variables designated by Roman letters from "a" to "z".

II MATHEMATICAL MODEL

COORDINATE SYSTEM

The project operations are carried out in a rectangular Cartesian coordinate system. With the help of the given coordinate system position of any point can be unambiguously defined.

Internal representation of data in the system is based on the right-hand coordinate system (in view "side" axis **X** is directed from the left to the right and axis **Y** – forward, axis **Z** – bottom-up). Further in the text this coordinate system will be referred to as the "**mathematical**" system.



GRID AND COORDINATE SYSTEM OF THE BODY LINES DRAWING

The lines drawing coordinate system (LD CS) is introduced to make the user's work convenient and is a coordinate system through which the user interfaces the system. It is set up with the help of the "**Settings/Grid**" command and may differ from the mathematical coordinate system in direction of the axes.

The LD grid is set up in the same menu item. This is the grid which is displayed on the system screen and used to match the points and display the sections. Positions of the LD grid sections are set up in the LD coordinates.

UNITS OF MEASUREMENT

The unit of measurement in the system represents a meter. The user can introduce the coordinate values with the random accuracy; the introduced coordinates are displayed in the prompt line with the accuracy of up to the decimal fraction of a millimeter.

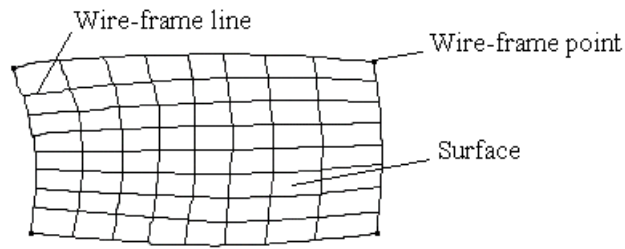
Some values (for example, thickness of a plate) can be given in millimeters in the relevant dialogues.

OUTLINE

The basic elements of the *SEA SOLUTION* system **mathematical model** are: a three-dimensional **point**, a three-dimensional **line** and a three-dimensional **surface**. The line starts and ends at the points. The lines can be interconnected at the points thus forming a closed contour. The surface is supported by this contour, the lines are the surface rulers, points are the surface angular points.



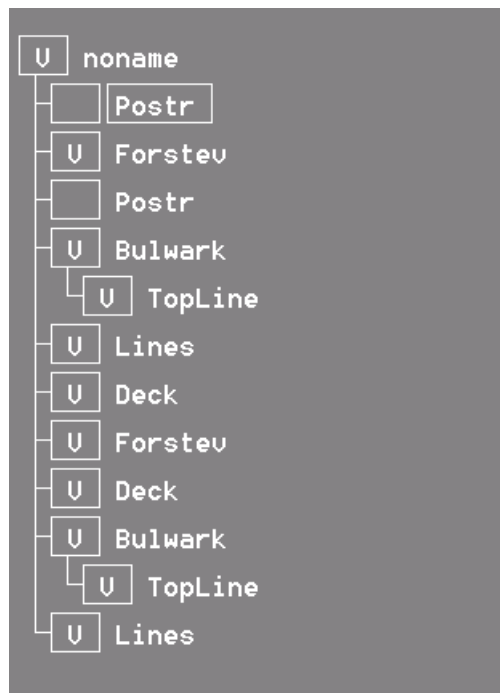
II. MATHEMATICAL MODEL



All components of the project, starting from the surface of the hull, decks, bulkheads and ending by the equipment, structure elements and the piping, are built up on the base of this model.

To perform compound constructions such as surfaces of revolution, line radius conjugation, surface conjugation (“fillets”), control elements called **drivers** are used. The drivers are also used for designing pipelines and the framework profiles.

Blocks combining different elements of the mathematical model are used for structuring the project elements in the *SEA SOLUTION* system. Each element belongs to a certain block, and to one block only. The blocks form a structure of “**nested**” blocks or a “**tree**” of blocks:



The drivers and blocks are the principal instrument for solving the tasks of general arrangement, hull structure and pipeline design.

Finally, there are elements intended principally for obtaining the project “out” documentation.

The *SEA SOLUTION* system allows introducing changes into the project at any stage of design development, and at this time all required relocation of the project constituent

II. MATHEMATICAL MODEL

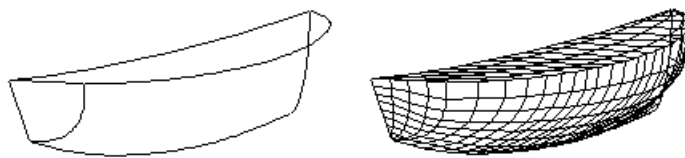
elements is made automatically. For instance, changing the hull geometry results in automatic relocation of the framework.

In this chapter we shall confine ourselves to dealing with but the points, lines and surfaces and only to that extent required for understanding general properties of the mathematical model.

To be concise, we shall refer to the mathematical model as the *SEA SOLUTION* **model**.

DESCRIPTION OF MATHEMATICAL MODEL

The hull surface design engineering, while using this mathematical model, consists in the following. At first, the lines forming a three-dimensional skeleton of the object are introduced. These may be buttock in CL lines, sheer lines at side, breaking lines and main frame line. Then the surfaces are “pulled” on this skeleton (the Figure shows the surfaces of the side, the deck and the transom). Finally, the lines and the surfaces are adjusted to obtain the required hull configuration (the configuration form is controlled by the cuts). At the same time, there is no need for matching the configurations as the hull one-piece surface is being designed and not single cuts (as it occurs on the manual lines drawing).

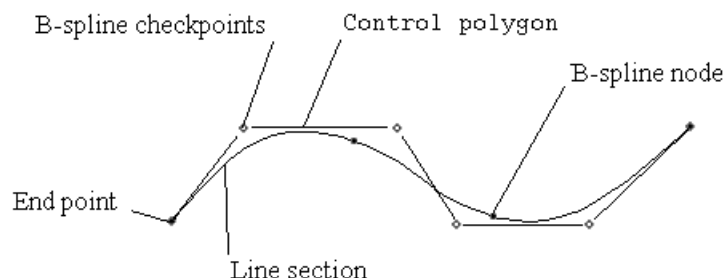


POINT

The point has three coordinates defining its position.

LINE

The line represents a smooth (doubly continuously differentiable as per the parameter) parametric curve in the three-dimensional space. It is introduced in the form of nonuniform cubic polynomial **B-spline**.



Such curved line represents a set of segments – **Besier** sections which are cubic parametric curves and which are interconnected at points called B-spline **nodes**. The number of Besier curve segments is less than the number of its checkpoints by 3.

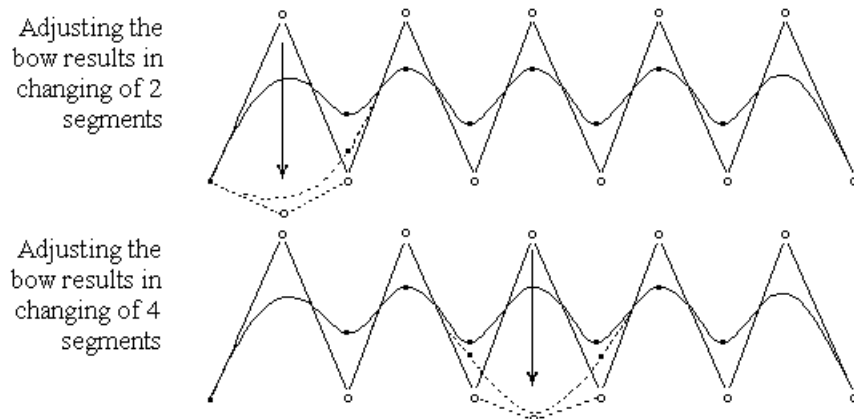
II. MATHEMATICAL MODEL

Position of any point on the line is defined by its parameter that changes monotonously and continuously along the curve.

A **direction** is defined for the line, i.e. its origin and end. Initial and end **points** of the line define its direction.

The B-spline is defined by the **control polygon**, to which according to some principle the curve with the following properties is correlated:

1. the control polygon must contain four B-spline **checkpoints** at the minimum (hereinafter referred to as the “**bows**”);
2. the initial and end points of the curve coincide with those of the control polygon;
3. the tangent at initial point of the curve is directed along the first segment of the control polygon and at the end point – along the last one;
4. the curve traces the control polygon form (in particular, the self-intersecting curve corresponds to the self-intersecting control polygon; if all the apexes of the control polygon are in straight line, the curve will coincide with this straight line);
5. the curve is contained in the control polygon “convex shell”, i.e. the curve dimension is knowingly not more than that of the polygon;
6. changing position of one of the control polygon apexes results in changing of maximum four segments of the curve;



7. the arcs and circles are approximated in the rough, the maximum radial deviation from the true arc can be 0.1 mm.

From the user's point of view, the control polygon is a means of the line form adjustment.

A line rests upon 2 **points**. A line starting and ending at one and the same point is not used and cannot be introduced. A line changes its form when position of the end point is changed.

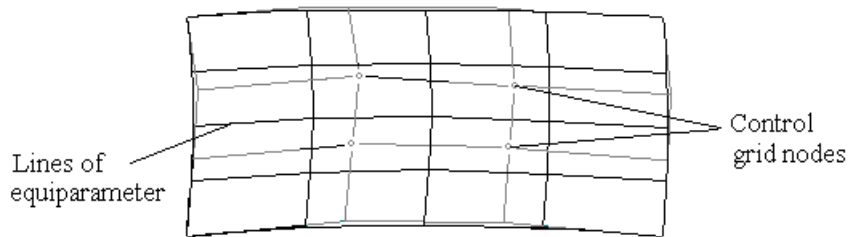
SURFACE

The surface-element is a smooth parametric **B-spline** surface. Its mathematics is similar to that of the **B-spline** curve, with correction for a two-dimensional case.



II. MATHEMATICAL MODEL

The surface is supported by 2, 3 or 4 rulers forming a closed contour. The closure is ensured if the surface angular point is common for two rulers. The surface changes its form when the form of the rulers is changed.



Lines of equiparameter and **equicut** give the idea of the surface form. The surface form can be controlled by changing the form of the rulers and position of the surface control grid nodes (bows).

The control polygon of the rulers define the number of the surface control grid bows and their position. If the opposite rulers have equal number of the bows, the surface grid along the relevant direction will have the same number of the bows. Otherwise, the number of the surface bows along this direction can be increased, but not more than the sum of the bows of the mentioned rulers.

DRIVER

The *SEA SOLUTION* system includes compound constructions such as a surface of revolution, line radius conjugation etc., and their automatic support during adjustment. The driver elements structurally are similar to the ordinary elements. They possess all the related topological dependencies and interconnections. They can be used for construction of other elements (lines, surfaces etc.), for object tie-in (geometrical and topological) as basic elements of fixation. Adjustment of the driver initial elements or the driver parameters results in automatic relocation of the driver elements.

REFERENCES

As stated above, a line changes its form when the end points change their position, a surface changes its form when the rulers are changed. Such dependency is realized by means of **forward** and **backward references** between the elements. For instance, the line has forward references to its end points, and these points have backward references to the line, the surface has forward references to the rulers and angular points, and the rulers and the points have backward references to the surface.

TOPOLOGICAL DEPENDENCY OF ELEMENTS

An element shall be referred to as the **topologically dependent** element on the other one (**reference element**), if it has at least one common point with the reference element, forward reference to the reference element and is changed as the reference element is changed. A line is topologically dependent on its end points, a surface – on the rulers and angular points.

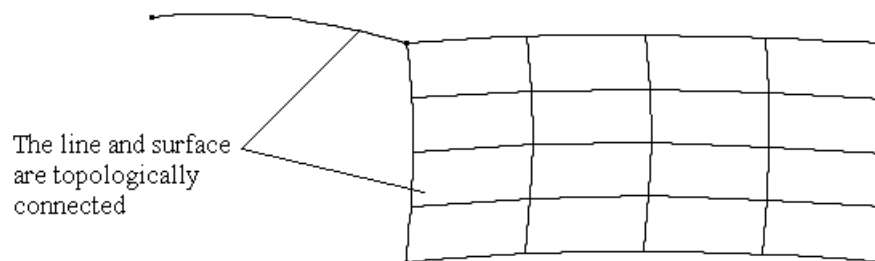
II. MATHEMATICAL MODEL

TOPOLOGICAL CONNECTION OF ELEMENTS

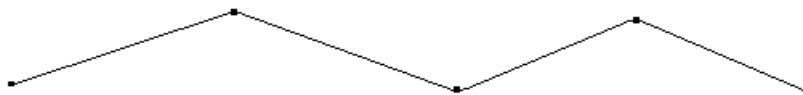
Two elements shall be referred to as the **topologically connected** elements, if there is topological dependency between them. A line and its end point, a surface and its ruler and the angular point are topologically connected.

Two elements shall also be referred to as the **topologically connected** elements, if there is an element common for them, on which these two elements are topologically dependent or which itself depends on these elements. Two lines are considered topologically connected, if they have a common end point. Two surfaces are considered topologically connected, if they have a common ruler or a common angular point. A surface and a line are topologically connected, if the line “comes” to the surface angular point.

Thus, the reference contour for the surface is closed if its lines are topologically connected. Such contour shall be referred to as the **topologically closed** contour.



A chain of lines shall be referred to as the **topologically connected** lines, if they are topologically interconnected except for the marginal lines.



DATA BASE

Elements in system SEA SOLUTION are stored in a uniform database. Each element has number - the whole positive number. Number is unique, in case of removal of an element his number cannot be appropriated to other element. Further number of an element we shall name his system name, or it is name.

THREE-DIMENSIONAL WINDOW. DISPLAYNG. SCALING

A **current three-dimensional window** (hereinafter referred to as the “**current window**”) is a certain volume in the project coordinate system, limited by rectangular parallelepiped.

The mathematical model elements are displayed only, if they fall within the current window. If a part of an element (a part of a line or a surface) falls within the current window, only this part of the element is displayed.

Original size of the current window coincides with that of the lines drawing grid.

The window size and position can be changed, thereby scaling the image display and removing unnecessary elements from the window.



II. MATHEMATICAL MODEL

LEVELS, COLORS AND BLOCKS. VISIBILITY OF ELEMENTS

Levels, colors and blocks are used to structure the project elements and to manage the element **visibility**.

The *SEA SOLUTION* system output levels are varieties of displayed objects: lines drawing grid, points, lines, a surface in equiparametric lines, surface cuts etc. The levels can be turned off making the relevant elements **invisible**, i.e. they will not be displayed, in DXF-file. Each element is specified by its color. The system uses 16 standard colors.

A block is integration of different elements with the purpose of their structuring. The *SEA SOLUTION* system applies the structure of “nested” blocks, or the block “tree”. See Chapter X for description of blocks and visibility management in a block.

CURRENT BLOCK. CURRENT COLOR

The system uses notions of a current block and a current color. Upon input all new elements fall within the current block. The input elements “change to” the current color.

ISOMETRY

The *SEA SOLUTION* system commands (entry, adjustment and etc.), with some exception, are executable during operation in isometry (this term shall be used to define the arbitrary axonometric mode).

OPERATING PLANE. OPERATING POINT. OPERATING POINT DEPTH

Mathematical model of the *SEA SOLUTION* system is a three-dimensional model. However, while entering a point by means of a marker, only two coordinates can be specified. To define the third coordinate, the **operating plane** is used.

Generally, in isometry the operating plane is always parallel to one of the base planes or to the screen plane. In views “**Front**”, “**Side**” or “**Plan**”, the operating plane is parallel at the same time to the relevant base plane and to the screen plane (unless the perspective is set up).

The operating plane always passes through the **operating point**. Controlling the operating point position allows controlling the operating plane position.

Coordinate of the operating point along the axis perpendicular to the operating plane shall be referred to as the **depth** of the operating point. In views “**Front**”, “**Side**” or “**Plan**”, this will be a coordinate along the axis directed “towards the screen depth”.

Accordingly, the operating plane shall be defined by its orientation and depth.

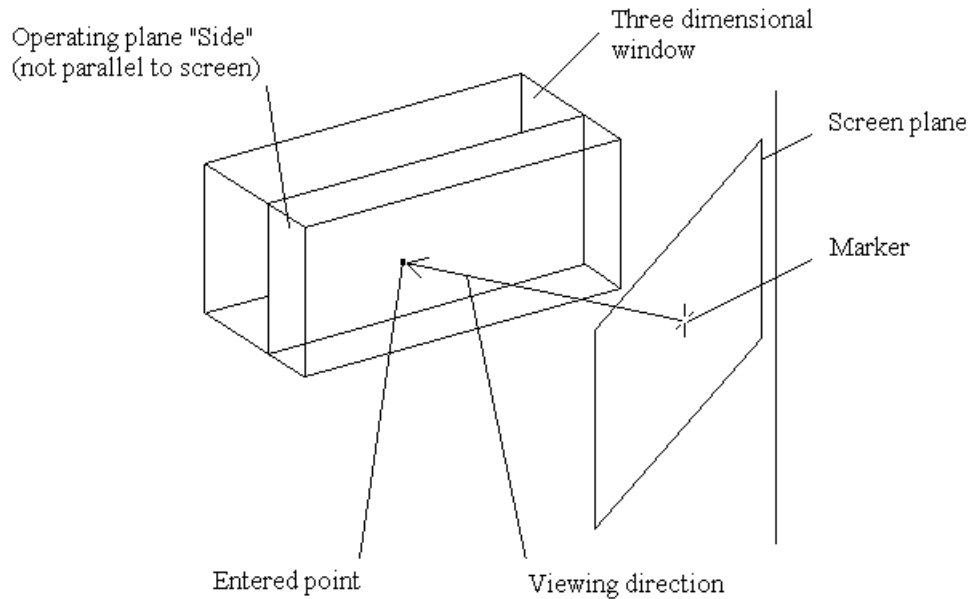
It is obvious that we can speak of the operating point depth and the operating plane only in respect of the operating plane “**Front**”, “**Side**” or “**Plan**”.

Hereinafter the term **operating projection** will be used meaning projection corresponding to the operating plane.

The point being entered proves to be the marker projection onto the operating plane towards the viewing direction, i.e. it is laid down upon the operating plane.



II. MATHEMATICAL MODEL



For instance, let the operating point have coordinates 10, 20, 30. In view “Side”, the operating plane is parallel to the CL plane (i.e. the operating plane is “Side”, its depth is equal to 20). Then coordinates **X** and **Z** of the entered point will correspond to the marker position, and coordinate **Y** will be equal to the operating plane depth and have the value of 20.

Let the operating plane “Plan” be set up during entering the point in isometry. Then coordinates **X** and **Y** will correspond to the marker position, and coordinate **Z** will have the value of 30.

The operating point always coincides with the last point entered. For instance, if in the preceding example the point is entered with coordinates 15.732, 26.496, 30, then the operating point will have coordinates 15.732, 26.496, 30.

The operating point and operating plane behavior will be discussed in the following chapters of Part Two using the specific examples.

OBJECT TIE-IN

The object tie-in allows using the geometry of other elements (tie-in objects) while entering the points, for instance, entering such points as a point lying on a line, the point of a line and the grid line intersection, the point of intersection of two lines of the grid and etc.

APPROXIMATION

The *SEA SOLUTION* system uses several methods for approximating the wire-frame lines or surfaces to the known in advance approximation points.



III. SERVICE COMMANDS

INTRODUCTION

The top menu contains control facilities a conclusion of elements of the project on the screen, management of movement of a marker on the screen, managements of an objective snap of points and some other service commands. In this chapter the most necessary commands are described.

Pay attention that the majority of commands of the top menu are "transparent", that is can be entered at performance of other commands.



GO

"Go" allows to draw again the image on the screen of the display. For performance of a command it is enough to press button "Go".

VIEW

The menu "View" is necessary for installation of the current view of object, and also for installation of parameters of prospect.

FRONT, SIDE, PLANE, ISOM

These items of the menu allow to establish the current view:

hull - YZ,
side - XZ, p
plan - XY
or an isometric.

If you want to set up a required view you should press the corresponding button of the menu, and then press button "Go".

RACURS

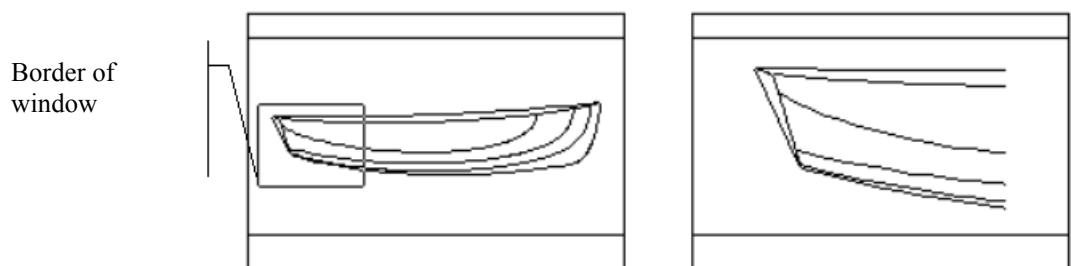
The item "Racurs" allows to change a foreshortening for an isometric projection. You should turn of object around of coordinate axes.

WINDOW

The menu "Window" contains control facilities in the sizes and position of the current window of the project. It allows to scale the sizes of the image of object and to exclude from work unnecessary elements (for example, at designing stern to set a window containing only stern). It will speed up search of elements.

NEW

The item "New" allows to set a new current window. For the task of a window it is enough to specify two his opposite corners and press button "Go".



PREV

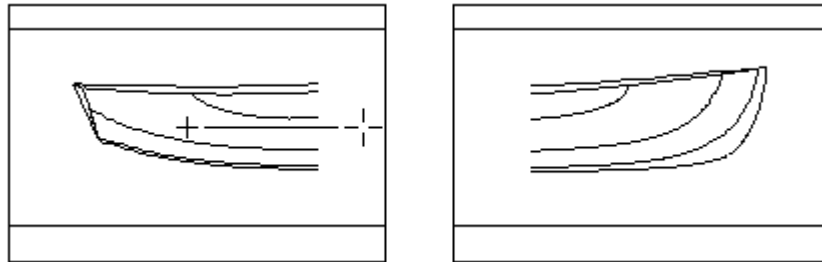
Allows to return to the previous window. The system keeps from two to five windows, since the initial window established by default. For execution of a command you should press button "Go".

DEFAULT

Allows to return to the initial window determined of dimensions of a grid of a body line drawing of the project. All kept windows (see the previous item) are removed. For execution of a command you should press button "Go".

SHIFT

Allows to move the current window to a new position, keeping the sizes and scale of the image. For execution of a command you should press button "Go".



←,→,↑,↓

Allows to move current window to the left, right, up, or down.

ALL

Allows to set up current window, which consist all visible elements of a project. For execution of a command you should press button "All" and then "Go".

LEVEL

The menu "Level" allows to operate visibility of elements of various levels (a point, a line, a surface). Elements are seen, if the suitable level is switch on.

If you want to switch on or switch of a level you should press or press back a suitable button of the menu and then press "Go".

POINT

You can see points if the level "Points" is switched on

LINE

You can see lines if the level "Line" is switched on.

SURFACE

You can see surfaces if the level "Surface" is switched on.

SECTION

You can see sections of surfaces if the level "Section" is switched on.

GRID

You can see "Grid" of body lines drawing which was set up in item "File/Grid" of lower menu.

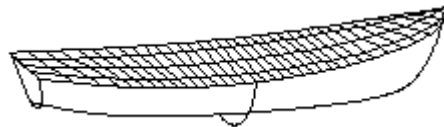
SECPLN

Secant plane ("Front", "Side" или "Plan") defines what kind of sections (frames, battocks, waterlines) appears on the screen on the view "Isom" if "Level/Sections" is switch on. For execution of a command you should press button "Go".

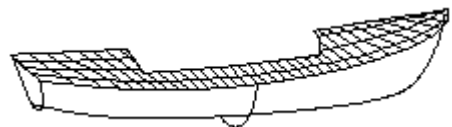
CUTTINGS

You can see cuttings on lines and surfaces if the level "Cuttings" is switched on.

Switch of



Switch on



SIZE

You can see angle and radial dimensions if the level "Size" is switched on..

DETAIL

You can see plates with their thickness if the level "Detail" is switched on.

IDENT

You can see text signatures to the points set by means of a command "Line/Text" if the level "Ident" is switched on.

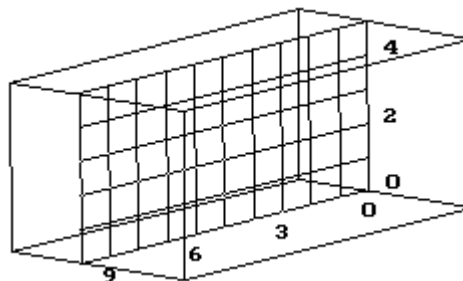
APPROX

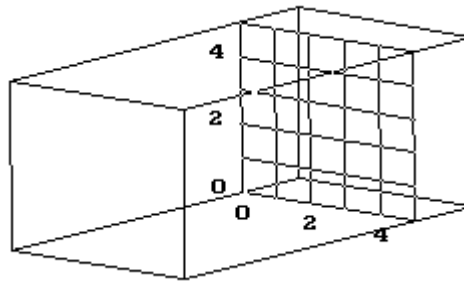
You can see approximate point if the level "Approx" is switched on.

WrkPln

The menu "WrkPln" allows to operate position in space of a working plane and a working point

The working plane is represented by lines of a grid of the body lines drawing, laying in a working plane





FILTER

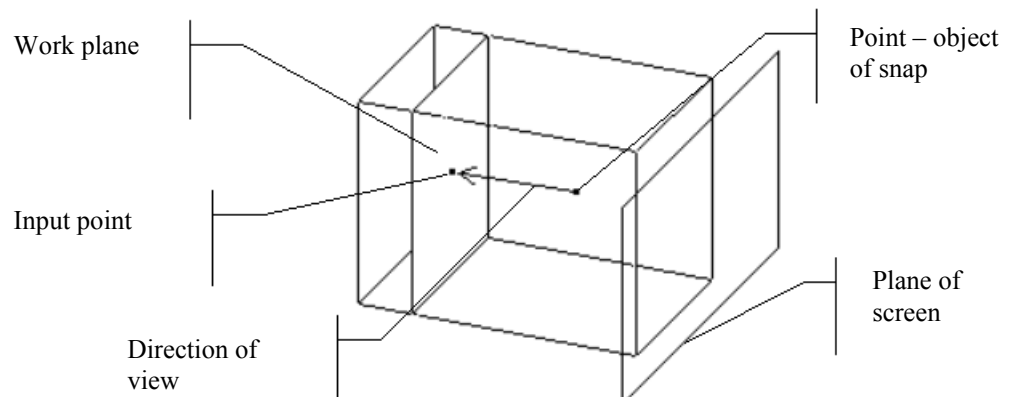
It is an object snap

PNT3D

Allows to combine a point with other point on 3 coordinates (points are coincided).

PNT2D

Allows to combine a point with other point on the current view.



POINT

This snap allows to begin or finish input of line in already existing points (the new point is not created).

Snap “Point” allows to attach a corrected point to other point. The corrected point is deleted, and lines, surfaces, etc., having direct references to this point, get references to a point of snap.

GRID

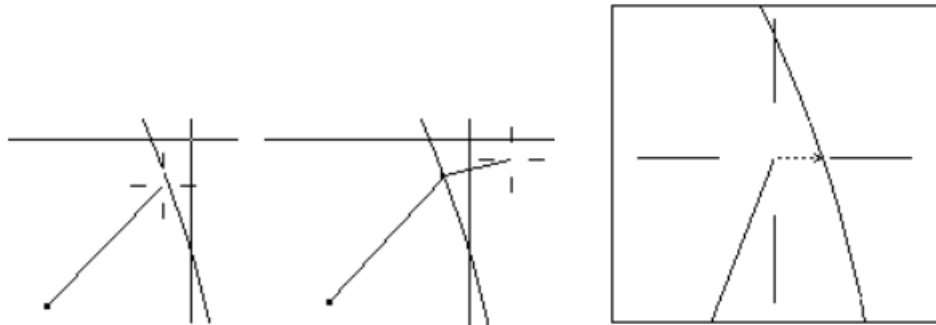
This snap allows to combine a point with a knot of grid of body lines drawing on the current view.

GRIDX, GRIDY, GRIDZ

These snaps allow to combine a point with a line of grid of body lines drawing.

LINE3D

Allows to combine a point with a line on three coordinates (i.e. the point will lay on a line).



LINE2D

Allows to combine a point with a line on the current

LINE

Allows to combine a point with a line on three coordinates, and establishes topological dependence of a point on a line (the point becomes hinged on a line, and a line - basic for this point).

SURF3D

Allows to combine a point with a surface on three coordinates (that is the point will lay on a surface).

SURFACE

Allows to combine a point with a surface on three coordinates and establishes topological communication of a point with a surface. The point becomes superficial, and a surface - basic for this point. The point receives type " fixed on a vector ", the projecting vector is perpendicular the current working plane.

X, Y, Z

Allows to receive a point with a preset value of one of coordinates.

For this purpose it is necessary to establish simultaneously a mode of snap to a line and this mode (two buttons should be pressed).

The system will give out inquiry " Fixed coordinate value: 0 " after input of a point. You should enter value of the fixed coordinate.

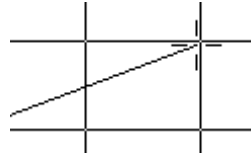
APPR

Allows to carry out a geometrical snap to approximate points.

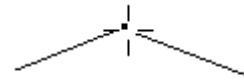
EXECUTION OF THE SNAP

For input of a point with a snap it is necessary to establish a required mode of a snap (to press corresponding buttons of the menu) before input of a point. Then to combine a cursor with object of snap (the object of snap should appear inside of marker) and to enter a point.

Snap to knot of the grid



Snap to point

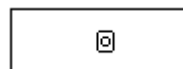


If an object of snap is not found or does not satisfy to conditions of snap, you will see the message: “Element not found”

Procedure of snap remains active, you can repeat an input of a point.

INDICATOR OF SNAP

Execution of snap is conformed to indicator of snap.



Snap to the grid on the number of frame

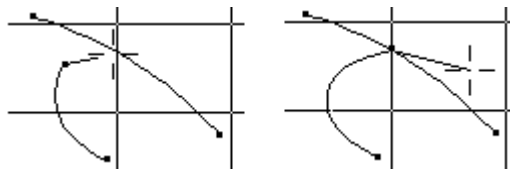
There is an opportunity of snap to lines of a grid under number of section in SEA SOLUTION. You should input a number of section in an inquiry line:

“Input point: x(7) - 0.15, y(2), 2”

The input point will lay from frame number 7 on 150 mm aside a negative direction of an axis X and will lay on buttocks number 2.

Combine snap

There is an opportunity of simultaneous snap to several objects, for example, a snap to a frame line and a line of a grid or a snap to a surface and knot of a grid. For this purpose it is necessary to establish a required combination of modes of snap..



WrkSrf

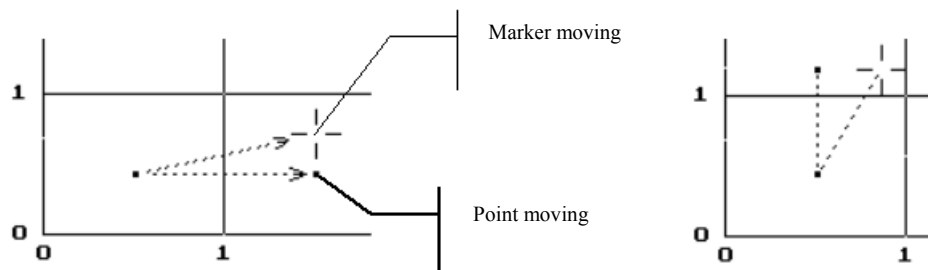
It is used for definition of a working surface. The working surface is used at input of surface lines and at an object snap of a point to a surface.

MARKER

At work frequently there is a necessity to move any point (the end of "a rubber thread" at input of the lines, a corrected point at correction, a point of snap of the block at work with blocks, etc.) in the certain direction, for example, vertically or horizontally or along any line. Opportunities of the menu "Marker" provide automatic keeping the set direction of moving.

ORT

The mode "Ort" provides moving a point in parallel lines of a grid of the current working plane. Except for a case of a working plane "Screen" on a kind "Isom" it means moving to in parallel corresponding axes of coordinates. The choice of a "vertical" or "horizontal" direction is carried out depending on an increment of coordinates of a marker.



DIRECT

The mode "Direct" provides moving a point along the straight line of the certain direction which is taking place through a starting position of a point. The direction of a straight line is set in item "Marker/SetDir".

DIRPOLE

The mode "DirPole" differs from previous that the straight line passes through separately set point - a pole. Position of a pole is set in item "Marker/SetPole".

DIRORT

The mode "DirOrt" provides moving a point perpendicularly with a straight line determining a direction and set in item "Marker/SetDir".

DIRLINE

The mode "DirLine" provides moving a point along any curve (there is a line - an element of mathematical model). Thus the output and for the ends of a line on some distance is possible also. The line is indicated in item "Marker/SetLine".

SETDIR

The item "SetDir" allows to set a straight line determining a direction of moving of a point in modes

"Dir", "DirPole" и "DirOrt".

Ways of the task the following:

- two points - a straight line goes through two points. It is necessary to enter two points on inquiry of the system;
- angle - a straight line goes through a point under the set corner of an inclination (in the current working plane). It is necessary to enter a point and a corner on the system inquiry.
- normal to projection of line - Normal to a projection of a frame line on a working plane. By inquiry it is necessary to specify a point on the line
- tangent to line - Tangent to a projection of a frame line to a working plane.
- normal to surface - Normal to a surface in the specified point.

In case of need at the indicate of required objects it is necessary to use an objective snap

SETPOLE

The item "SetPole" allows to enter a point - a pole for moving a point to a mode "DirPole".

SETLINE

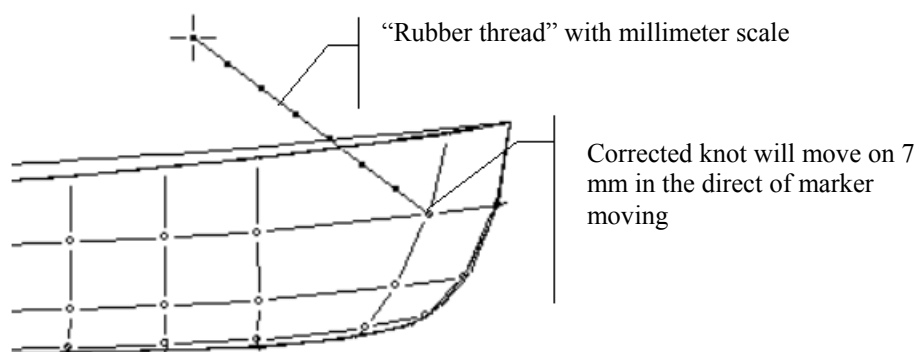
The item "SetLine" serves for the indicate of a line for moving a point in a mode "DirLine".

DIR2D, DIR3D

Moving of a point to the above-stated modes can be carried out either on 3 or on 2 coordinates, that is at moving "Dir3d" the point coincides with a line of a direction on 3 coordinates, and at moving "Dir2d" the point coincides with a line of a direction only on the current view, keeping one of coordinates constant. As a matter of fact, moving on 2 or on 3 coordinates to similarly geometrical snap to line on 2 or on 3 coordinates.

SCALE

Allows to carry out small moving of a point (about millimeter)



Select

Allows to execute some design commands with several elements simultaneously (for example to remove group of lines, not specifying consistently each line).

ELEMENT

WINDOW

Element has to completely got in some three-dimensional window set by the user by inquiry of system

CROSSING

In this way it is enough for the instruction, that in a set window the part of an element got.

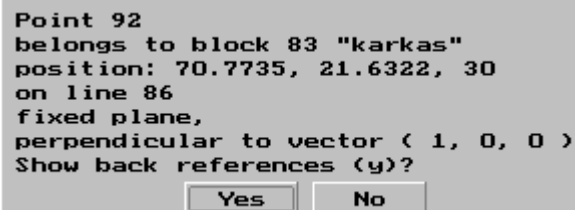
BLOCK

For the instruction of the block in this way it is enough to specify any element of this block by inquiry of system..

Tools

INFO

Allows to receive the information on elements of mathematical model. The necessary element is underlined or a marker in a working window, or input of a system name in a line of inquiry.



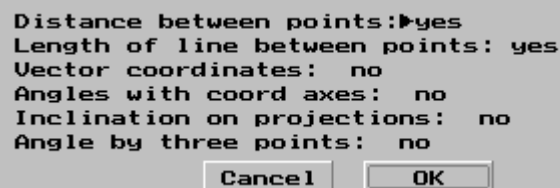
Point 92
belongs to block 83 "karkas"
position: 70.7735, 21.6322, 30
on line 86
fixed plane,
perpendicular to vector (1, 0, 0)
Show back references (y)?

Yes No

- **Point 92** – type and name of element;
- **belongs to block 83 “karkas”**
- **position: 70, 21, 30** - coordinates;
- **on line 86** – type of point and reference to basic element
- **fixed plane** – type of fixation;
- **perpendicular to vector (1, 0, 0)**
- **Show back references (y)?**

DIST

Allows to measure distance between two points, corners of an inclination of lines, lengths of lines, etc.



Distance between points: yes
Length of line between points: yes
Vector coordinates: no
Angles with coord axes: no
Inclination on projections: no
Angle by three points: no

Cancel OK

- **Distance between points: yes**



- Length of line between points:
- Vector coordinates: no
- Angles with coord axes: no
- Inclinations on projections: no
- Angle by three points: no

After that by inquiry of system it is necessary to specify two points (at measurement of a corner on three points - three). At the instruction it is necessary to use in case of need an objective snap (for example, at measurement of length of a line between two points belonging to this line, it is necessary to use a binding "Point", becoming attached by the ends of a line or points laying on it or to use the coordination on a line "Line").

After input of first 2 points the system gives out the required information or in the center of the screen, or in a line of messages.

```
Distance = 80.8617 m,  
Line length = 86.1173 m,  
Vector:  
Dx = -75.1587 m,  
Dy = 0.0000 m,  
Dz = -29.8293 m,  
Angles with axes:  
X: 158.353 grad,  
Y: 90.000 grad,  
Z: 111.647 grad,  
Angles on projections with axes:  
Front - with Y: -90.000 grad,  
Side - with X: -158.353 grad,  
Plan - with X: 180.000 grad
```

OK

```
Angle of points 1 - 2 - 3:  
In space: 68.638 grad,  
On front ( YZ ): 68.638 grad,  
On side ( XZ ): 68.638 grad,  
On plan ( XY ): 90.000 grad
```

OK

MODIFY

UNDO

You can cancel last execution command using menu "Undo" if you do not satisfied of the result.



IV. DESIGN COMMANDS

INTRODUCTION

Design commands of system " Sea Solution " is described in this chapter: loading of the project, the task of the user system of coordinates and grids of the body lines drawing, input of lines and surfaces, correction and removal of elements.



FILE

There are commands of menu “File” in this chapter. Commands of menu “File” allow to create a new model, open file with an exist model, set coordinate system and output of the system.

OPEN

This command allow to create new model or load exist model. Процедура состоит в выборе нужного файла или задании нового имени файла в окне выбора файлов, детально описанном в Приложении А..

EXIT

Close the system “Sea Solution”.

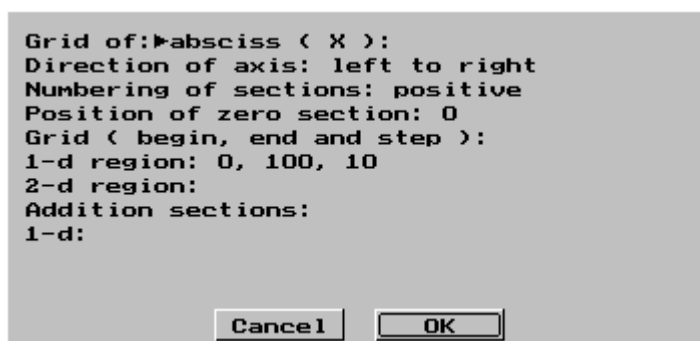
SAVE

Allows to save file

GRID

“Grid” allows to set coordinate system and grid of body lines drawing. Grid has following dimensions 100 x 50 x 50 meters and interval between sections 10 meters on each axis.

You can set coordinate system and grid of body lines drawing in dialog box.



Dialog box consist of following:

- **Grid of: X** – choice of a coordinate axis;
- **Direction of axis: left to right** – choice of a direction of a coordinate axis;
left to right or **right to left** – from left to right or from right to left for axis X;
from you or **to you** – for axis Y;
down to up or **up to down** – for axis Z.
- **Direction of sections numbering: positive** – allows to set up direction of numeration of sections, which can coincide with direction of appropriate coordinate axis ore be opposite.
positive - direction of numeration of sections and direction of numeration of coordinate axis are coincided;
negative - direction of numeration of sections and direction of numeration of coordinate axis are opposite
- **Sections (begin, end and step):**

in 1-d region: 0, 100, 10 – laying out of the grid. You can set to beginning, end and step of division.

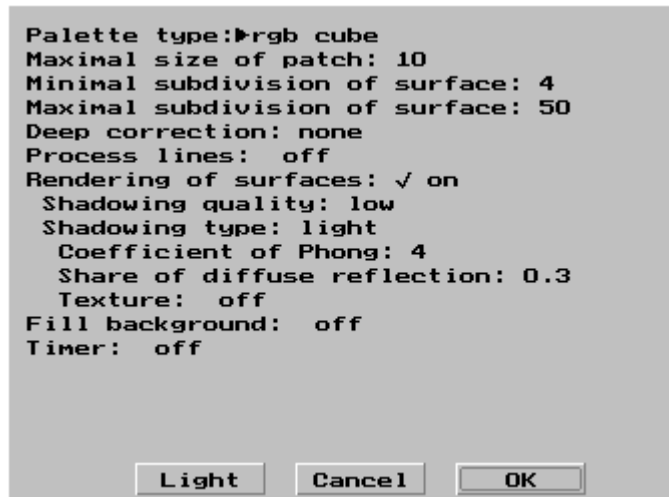
- **Additional sections:**

1-d: 0 – coordinates of additional sections.

Tools

REND

Rendering allows to receive realistic representation of projected object or his part which have been cut out by the current window, in the established racurs on the screen of the display.



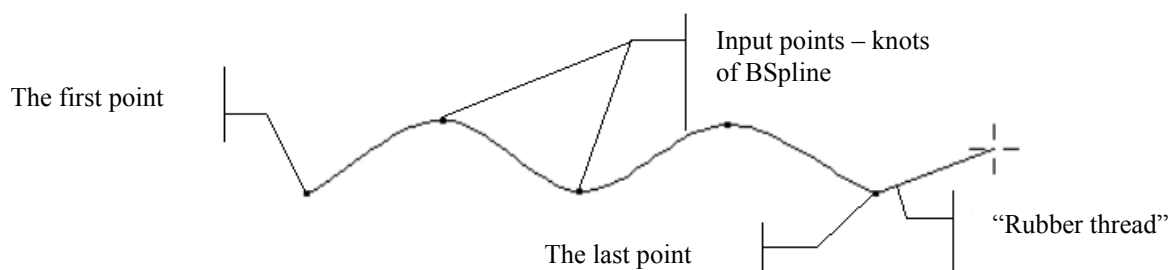
LINE

“Line” commands are used for input of framework lines.

SPLINE

If you want to input lines of “Spline” way you will input knots of Bspline

Thus the line goes through input points.



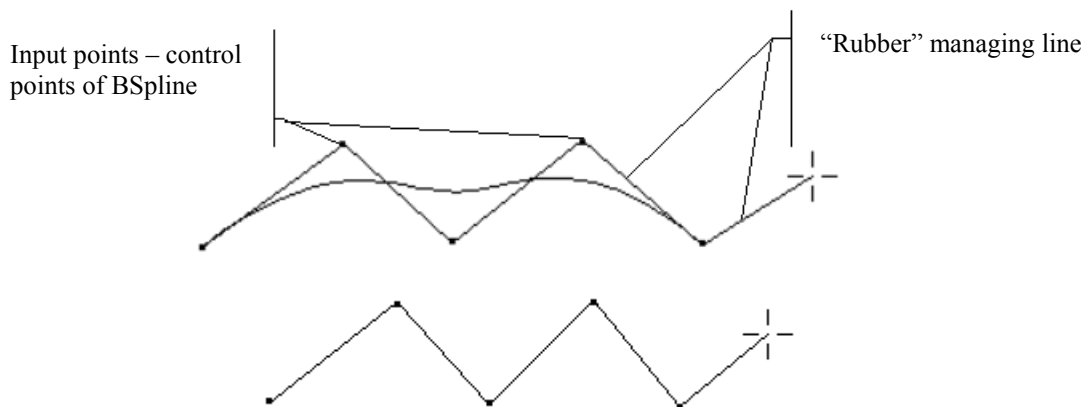
BSPLINE

If you want to input lines of “BSpline” way you will input control points of Bspline



LINE

Method inputs **“Line”** is a special case of **“Spline”**. It allows to input lines which consist of no more than one Bessie segment. It is convenient if you want to input a chain of straight lines



BEZIER

Method inputs **“Bezier”** is a special case of **“B-spline”** (you will input 4 points of management line).

SURFACE

The command **“Surf”** is used for input of surface lines. Procedure of input looks as follows.

You should indicate a work surface in item **“WrkSrf”** of upper menu. After that you should press button **“Surf”** and you will input a surface line.

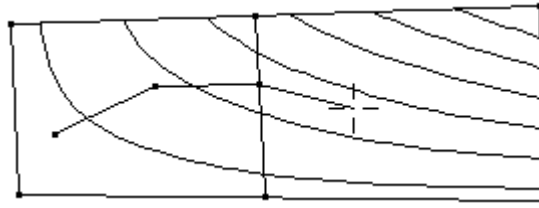
The pressed position of the button speaks that surface lines are entered.

Final points of a surface line receive type "fixed on two coordinates".

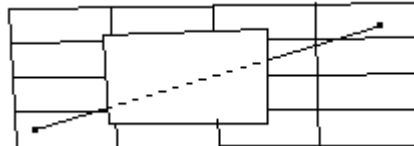
You can use object snap during input points of a line.

If you input a chain of surface lines it is possible to pass to other surface, which is connected with the work surface, without interrupting of input.

For this purpose it is necessary to enter a final point of a line with a topological snap to a boundary line or an angular point of a working surface, then to specify a new working surface and to continue input of a line.



If the entered surface line is passed through cutting on the working surface, you will see a cutting on the line automatically.



CROSS

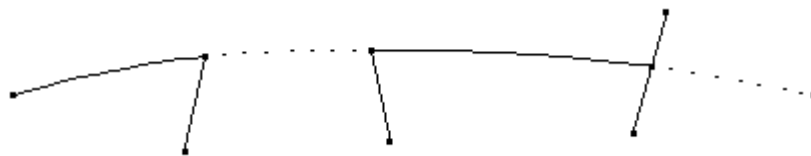
Allows to enter a line of crossing of two surfaces. For input of a cross line it is necessary to specify crossed surfaces by inquiry of system. If lines of crossing of surfaces more than one, will be entered all lines of crossing.

If a cross line is passed through cutting on the working surface, this line will be cut automatically.

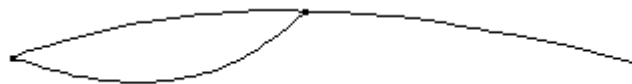
Allows to enter also lines of crossing of two drivers or the driver with a surface.

CUT

Allows to set cutting on a line between two points belonging to this line.

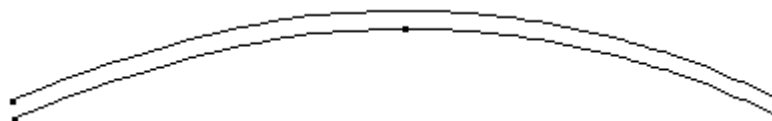


For the task of cutting it is necessary to specify two points which are limited cutting on an inquiry of the system. If the both points belong simultaneously to two lines, the system gives out additional inquiry to specify the necessary line



DIVIDE

Allows to create two lines with the general final point, precisely repeating the form of an initial line. Thus final points of an initial line become final points of new lines.



For input of lines you should to specify an initial line on the system inquiry " Select line " then you should to enter a point of division (including object snap) on the system inquiry " Input point: 0, 0, 0 "

If the initial line has no return references and it is not an element of the driver, it can be removed by inquiry: " Delete old line (y)? ".

POINT

Allows to enter a point of crossing of two lines or line with a surface. For input of a point it is necessary to specify on of lines or a surface and then the second line on inquiry of the system.

The point of crossing will be constructed, if the specified elements are really crossed

If crossing is not unique, all points of crossing are entered.

SIZE

This command provides the task of the linear, angular and radial sizes. You should choose type of the size and its parameters in a dialogue window.

Then it is necessary to indicate points and lines determining the size and also a point of snap which defines position of a dimensional line.

The entered sizes belong to the current block and are painted in the current color.

Thera three trypes of size:

LINEAR SIZE

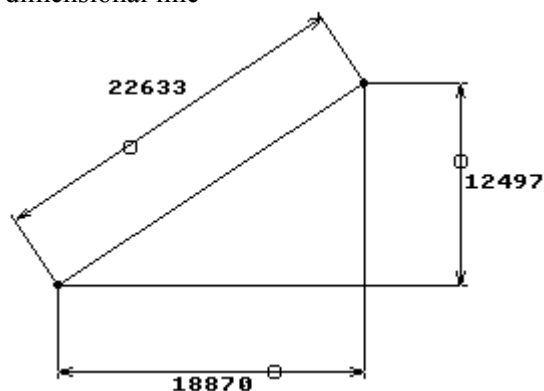
Defines distance between two points along the certain vector. The dialogue window contains the following parameters:

- **Type: Linear Size**
- **Direction: auto orthogonal** - parameter determining a direction of a vector along which the size will be measured:

vector - vector can be set in an appearing additional line

auto - vector always coincides with a direction from one point to another

auto orthogonal - vector will be defined, how in a mode "auto", or will be parallel to one of axes of coordinates, depending on a relative positioning of reference points and a point of snap of a dimensional line



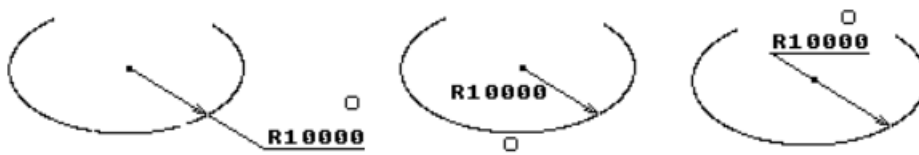
RADIAL SIZE

Defines radius of a projection of a basic line on the certain plane in the location of a reference point which should lay on a basic line. The dialogue window contains the following parameters:

- **Type: Radial Size**
- **Auto detect plane: yes**

If you choose "no", you should set up the normal to this plane (corresponds to the current working plane) in an additional line of dialogue Plane: 0, 1, 0

If you answer "yes" the plane gets out automatically.



ANGLE SIZE

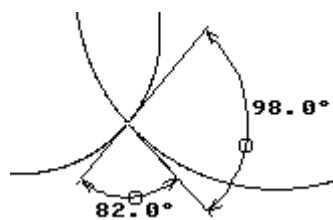
Defines a corner between projections of tangents to two basic lines in reference points laying on them on the certain plane. The dialogue window contains the following parameters:

- **Type: Angular Size**
- **Auto detect plane: yes**

If you choose "no", you should set up the normal to this plane (corresponds to the current working plane) in an additional line of dialogue Plane: 0, 1, 0

If you answer "yes" the plane gets out automatically.

The dimensional line passes through a point of snap.



ARC

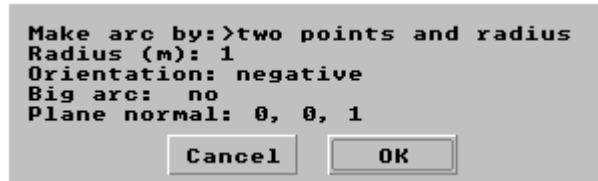
Allows to draw arcs on any projection.

There are several types of draw:


- arc of the set radius which goes through two entered points;
- arc which goes through three entered point;
- arc of the set radius with the center in an entered point;


If the option of input of superficial lines "Line/Surf" is included, the arc is projected on the current working surface. Otherwise the space line is entered.


A dialog window consists of several lines: Диалоговое окно содержит следующие строки:




- **Make arc by: two points and radius:**
two points and;
three;
center and radius
- **Radius (m): 1;**
- **Orientation: negative** – orientation of arc (for the first way);

negative


positive

- **Big arc: no** – arc with a smaller or big central corner (for the first way);




- **Plane normal: 0, 1, 0** - normal to a plane of a projection on which the line should have the form of an arc (for first of two ways)

SURF

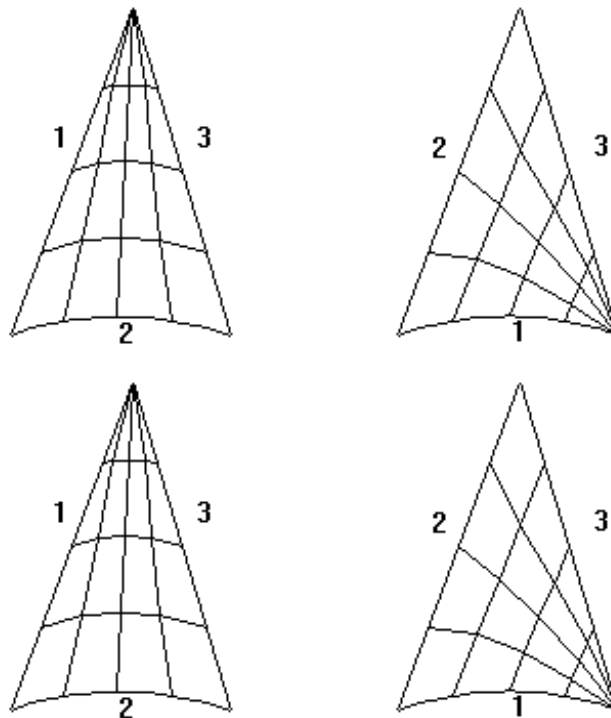
MAKE

If you want to create surface you should indicate 2, 3 or 4 lines, which are formed a topological closed contour, on system enquiry **“Select lines and press Enter”**.

If one of elements is invisible, you will see a system **“Level, color or block is off”**. You can continue input, but you can't see input surface. You can see surface if you switch on level **“Surface”** (**“Level/Surface”** in upper menu).

An input surface will belong to a current block and will has current color.

You should indicate 3 lines which formed a surface in following way:



In this case it is easier to get a necessary form of surface.

DELETE

It is necessary for delete of elements of mathematical model.

DELETE OF LINE.

If you want to delete line you should press button **“Line”** and on system enquiry **“Select element:”** indicate elements which you want to delete

You can't delete line if this line is a border of a surface. First of all you should delete the surface. If you delete line its ending points are deleted automatically.

DELETE OF SURFACE

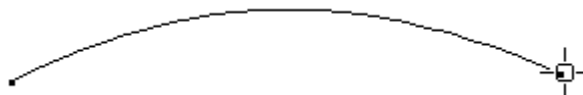
Surfaces are deleted as well as lines.

EDIT

“Edit” is used for edit of elements of mathematic model.

EDIT OF POINTS

After that you have pressed button **“Edit”**, you will see a system enquiry **“Select element or driver to edit:”**. You should indicate a point for edit. You will see a special symbol in the point and you will see enquiry **“Input point: 0, 0, 0”** in enquiry line.



If you move a cursor, an indicator of point will move behind the cursor. You should move cursor in necessary position and input point (press **“Space”**).

IV. DESIGN COMMANDS

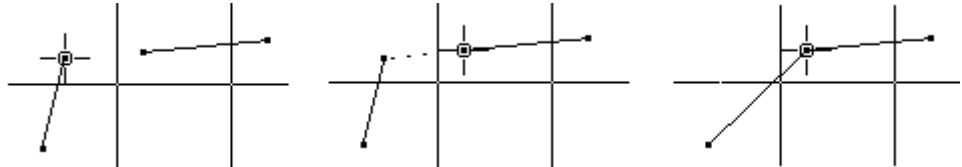
You should press **"Esc"** or indicate other element if you want to conclude of edit of this point.

During point corrections you can use all possibilities of input of point: object snap, work in isometric etc.

You can try to execute a topological snap point to other point.

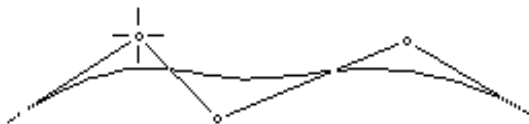
Indicate point

Move this point to object of snap and press "Space"



EDIT OF LINES AND SURFACES

You should indicate an edit line on system enquiry **"Select element or driver to edit:"**. You will see a management line.



You should indicate an edit knot on system enquiry **"Select knot to edit"**. You should move this knot in necessary position and input point on system enquiry **"Input point: 0, 0, 0"**. After that you should edit other knots. If you want to complete editing, press **"Esc"**.

Editing of surfaces is similar to line editing.

DIALOG IN EDIT

You should firstly indicate an element for edition after that input a necessary command.

STRGHT

This command allows to straighten a part or all managing line (in space or with preservation of one of projections), or a number of control points of a surface. It leads to straightening of a corresponding part of a line or a surface.

Straightening of a line

You should indicate an editing line and then press button **"Straight"**. You will see a following dialog box on the screen:



- **Area of action: all line:**

all line – all line is straightened;

partial - a part of managing line between two knots is straightened.

- **Saved projection: global straight** – saving projection:

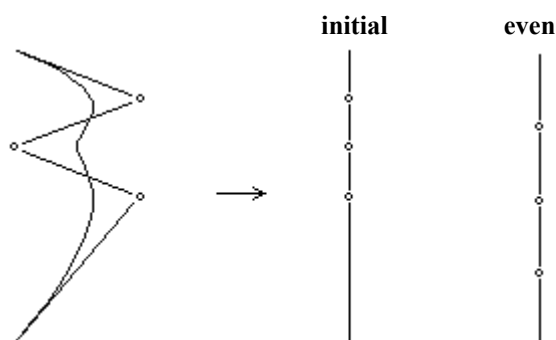
global straight – without saving (in space);

front, side, plane – with saving projection. Line will be straightened on the current view. If the current view is “**Isom**”, the line will be straightened on the work projection

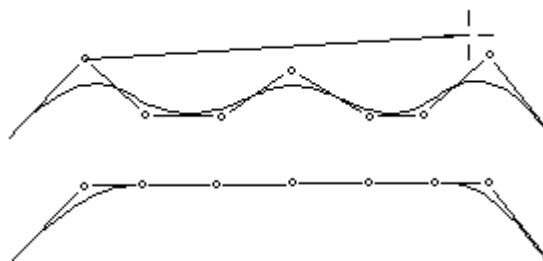
- **Knots distribution:**

even

initial.

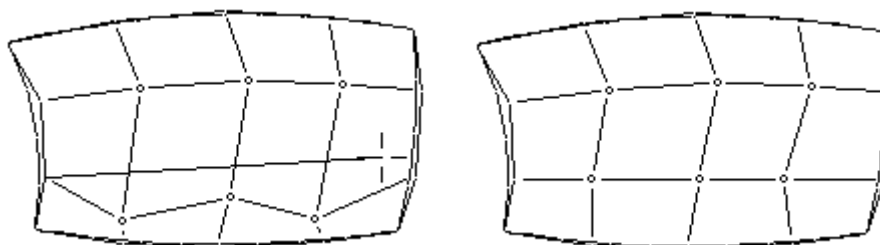


If you choose “**Area of action: partial**”, you should indicate 2 knots which are limited a straight part of managing line on system enquiry.



Straightening of a surface

A straightening of knots of a surface is same as a straightening of knots of a line, but you should always indicate scope.



JOINT

For smooth transition of one frame line in another it is necessary to provide coincidence of directions of tangents to lines in the general point.

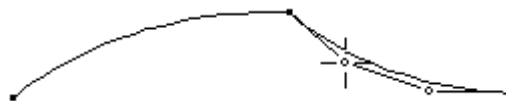
For this purpose adjacent parts of managing lines should lay on one straight line



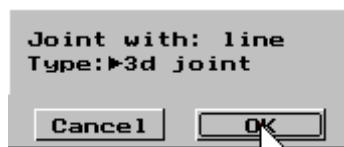
Let's name a line or a surface to which the corrected line or a surface is joined, object of joining

Join of frame lines

Indicate an editing line. Indicate an editing knot.

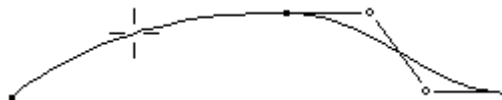


Press button **“Joint”**. You will see a dialog box on the screen:



- **Joint with: line** – method of join:
 - line** – join with line;
 - two lines** – join with two lines
 - surface** – join with surface
- **Type: 3d joint** – type of join:
 - 3d joint** - Tangents to lines in the general final point will be parallel in space;
 - front, side, plane** - The tangent to a corrected line will lay in a plane of the chosen projection and is parallel to a projection of a tangent to a line of joining.

If you want to join line with line (**“line”**) first of all you should choose method and type of join. After that you should indicate line on the system enquiry: **“Select jointing line:”**.



COLOR

You can set up a current color of lines and surfaces.

You can re-color existing lines and surfaces using this point of menu. You should indicate elements on the system enquiry **“Select element to change color”**.

V. DRIVERS

INTRODUCTION

There are complex constructions such, as a surface of rotation, interface of lines in radius, etc., and their automatic support are realized in system SEA SOLUTION. For management and supports of constructions are used drivers (item “**Driver**” of lower menu).

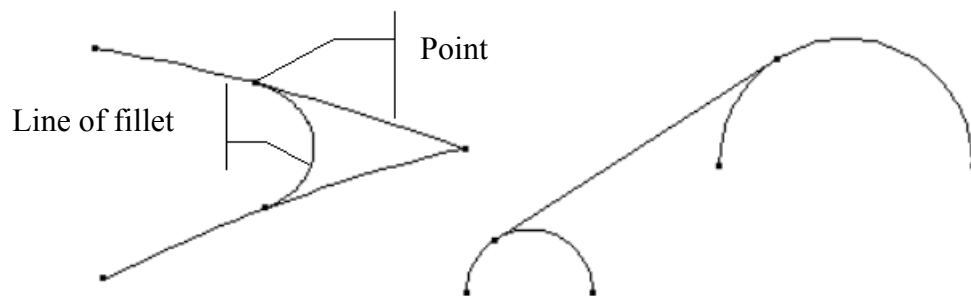


Strip

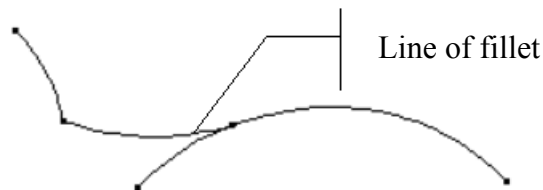
Allows "to disband" the driver or his separate elements. Disbandment of the driver stops his existence as the structure, elements of the driver become independent elements..

Fillet

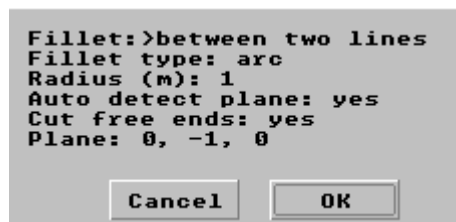
Allows to receive smooth interface of two lines by the line having the form of an arc or a straight line on a certain projection.



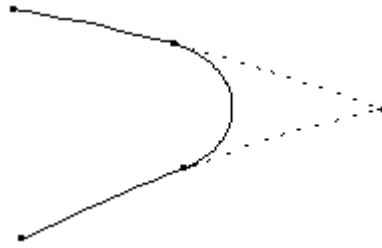
Allows to receive also "interface" of a point to a line an arc or a direct line.



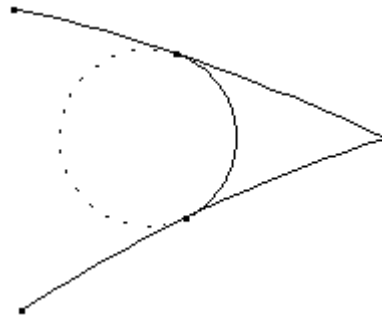
Dialog window "Driver/Fillet" consist of following lines:



- Fillet: between two lines:
 - between two lines
 - from point to;
 - from point on line to another line
- Fillet type: arc:
 - arc
 - straight
- Radius (m): 1
- Auto detect plane: yes
- Cut free ends: yes

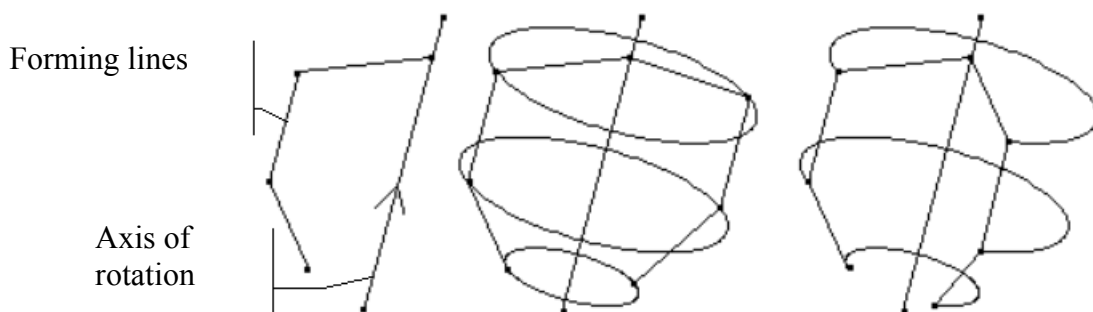


For reception of interface of two lines " between two lines " it is necessary to specify by inquiry of system lines. From two possible arcs is made smaller:



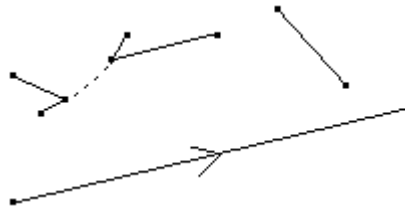
Rotate

Allows to construct the surfaces formed by rotation of forming lines around of an axis on any corner. It is used at designing tanks, elements of the general arrangement and the ship equipment.



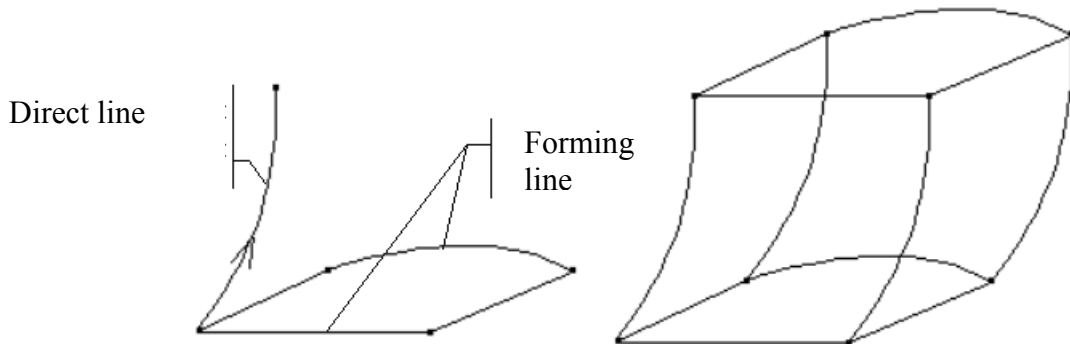
For performance of construction it is necessary to specify two points determining an axis of rotation, or a frame line by inquiry of system (the axis is defined by its final points). Then by inquiry of system to set size of a corner of rotation (clockwise concerning the specified direction of an axis). The corner of rotation can be negative, but should not be more than 360 °. Then to specify forming lines and to press "Enter".

Final points of forming lines can be any type. Forming lines can be not connected topologic among themselves. Presence of cutting on a forming line leads to formation corresponding cutting on a surface of rotation



Extrude

Allows to create surfaces which are formed by extrude of forming lines along the directing line. It can be used for designing a cylindrical insert of the hull, elements of hull construction and the general arrangement.



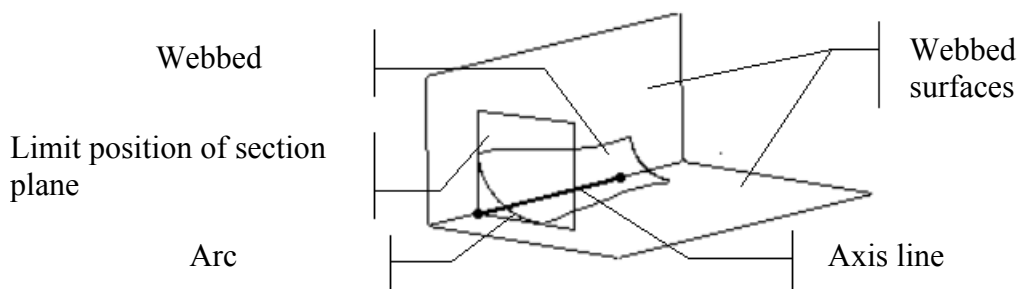
First of all you should to specify the directing line. The direction of extrude coincides with a direction of a directing line. If you want to change a direction of extrude it is necessary to specify directing line repeatedly. After that you should press **"Enter"**.

Webbed

Allows to make smooth interface of two surfaces by means of the third surface - surfaces of interface.

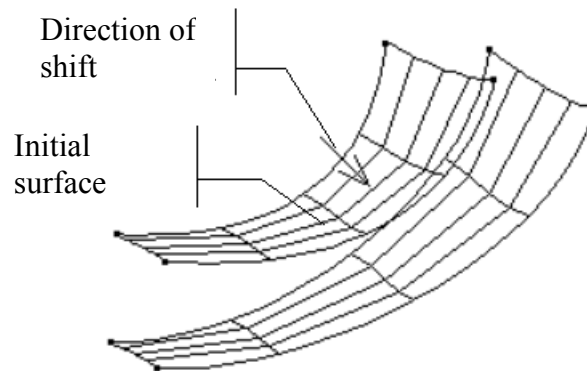
First of all you should specify wedded surfaces

The interfacing surface is created so, that its section any plane crossing an axial line perpendicularly to it, looks like an arch. In points of the ends these arches are smoothly joined with webbed surfaces. The arcs corresponding to the beginning and the end of an axial line, are edges of an interfacing surface.



Offset

Allows to create a surface, offset of other surface.

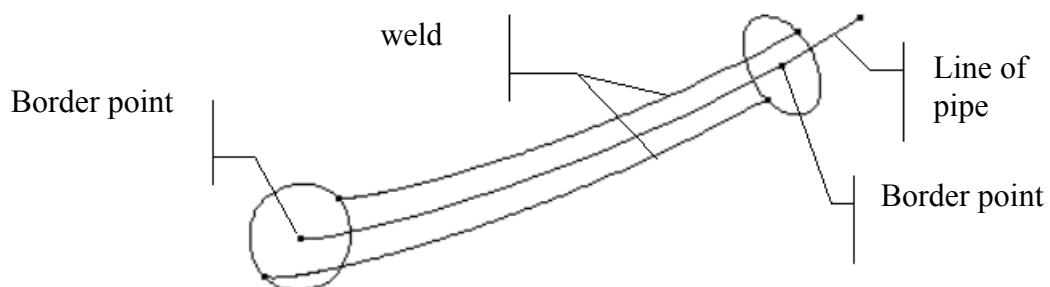


It is necessary to specify an initial surface by inquiry. After that you will see on the screen the arrow specifying an initial direction of shift.. Then by inquiry " Input shift (m): " it is necessary to set size of shift an offset surface. At positive value of size of shift the direction of shift coincides with initial, at negative - is opposite.

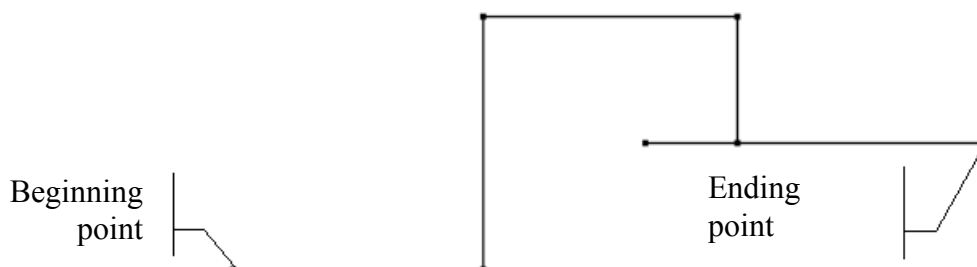
Pipe

This driver is used for design of pipelines.

An initial elements of a pipe are the frame line representing a theoretical line of a pipe, and the frame points limiting a pipe. Boundary points can be any points belonging line.



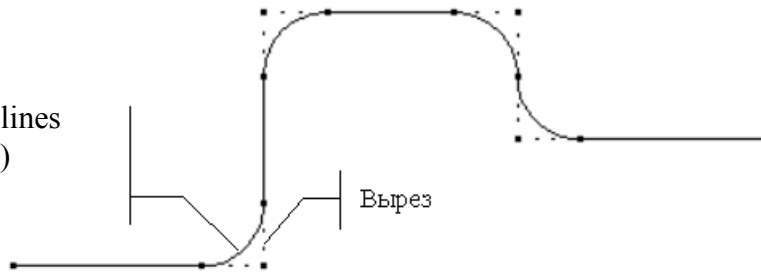
Means of item "Pipe" allow to project as separate pipes (a line from one line), and sites of pipelines of constant diameter with complex lines from several lines.



Points of a line should be topological connected.

The lining of a line is carried out by system automatically. Thus in places of a bend (in points of demolition of a line) interfaces of lines (Fillet) automatically are under construction. The "Free" ends of lines are cut automatically.

Fillet of two lines
by arc (Fillet)



Dialog window consists of following lines:

```
Diameter (m): 0.05
Trace:>filleted pipeline
Fillet radius (m): 1
Cut free ends: yes
```

Cancel

OK

- **Diameter (m): 0.05;**
- **Trace: filleted pipeline:**
pipe:
pipeline:
filleted pipeline:
- **Fillet radius (m): 0.1.**
- **Cut free ends: yes.**

It is necessary to specify an initial and final points of a line by inquiry for creation of the pipeline. If there are branching it is necessary to specify the necessary line of a pipeline by inquiry of system.

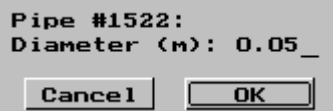
Edit of pipe

You can change a diameter of pipe:

DELETE

Driver

The forming element of the driver can be removed only in structure of the driver. Removal of drivers is carried out in item "Delete/Driver" the bottom menu. For removal of the driver it is necessary to specify his any element.



VI.BLOCKS

INTRODUCTION

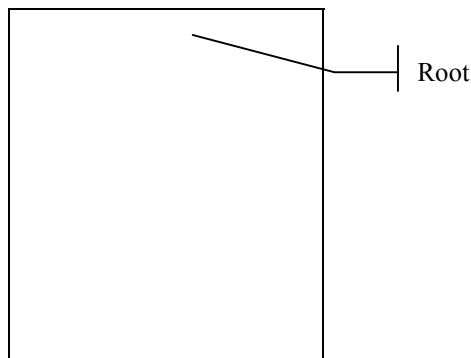
Blocks are used for structurization of the project in system SEA SOLUTION. The block is an association of geometrical elements - points, lines, surfaces, drivers, the sizes both sheet details - and other blocks. Each geometrical element necessarily belongs to block.



The basic concepts

Blocks also can belong to other blocks, forming structure of the enclosed blocks, or "tree" of blocks. The blocks which are not belonging to other blocks, are root.

The blocks which do not belong to other blocks, are root.



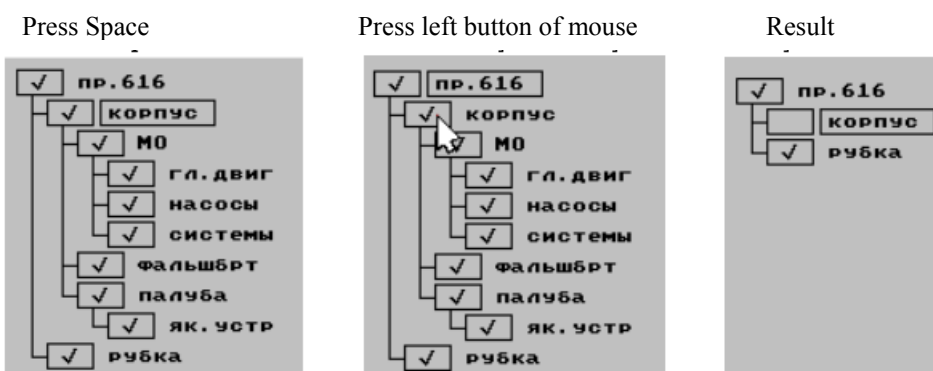
The menu "Block" allows to operate visibility of blocks and elements inside the block and to carry out some other actions with blocks

Tree

Allows to include and switch off blocks (management of visibility of blocks), and also levels (Level) and colors inside the block (management of visibility inside the block) to change an arrangement of blocks in a tree to rename blocks and some other operations

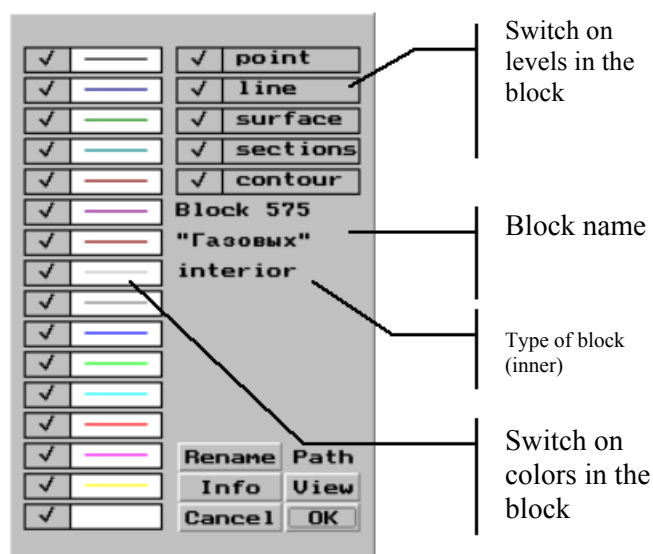


1. You can switch on/ switch off block



All elements of block will invisible.

2. For management of visibility inside the block the dialogue window of attributes of the block is used. For a call of a window of attributes it is necessary to press button "Attr" of a window of blocks, a key "Ctrl+Enter" or repeatedly to specify the current marked block a marker.



You can switch in/ switch off colors and levels inside the block

VISIBILITY OF ELEMENTS

It is necessary to switch on corresponding level in Level of upper menu.

AllOn

The mode "AllOn" allows to make seen all elements of the project. The mode is included at the pressed position of the button. After inclusion this mode it is necessary to press button "Go ".

Entry

Allows to appoint any block "a point of an input" in a tree of blocks. All blocks which are not belonging directly or indirectly to this block, become invisible (elements of these blocks are not displayed).

To return a tree of blocks to an initial condition, it is necessary to press repeatedly button "Entry" and in a dialogue window to press "Esc" or "Cancel".

Current

Allows to establish the current block. In the current block are located at input new elements of the project (a point, a line, a surface, blocks, etc.).

LOWER MENU. BLOCK

The instruction of the block or group of elements

Add

Allows to create the internal block in the current block. For creation of the block follows by inquiry of system " Block name: " to enter the name the block (no more then 8 symbols)

The new block can be established current if to answer yes on the inquiry of the system

“Do you want this block be current (y)?”.

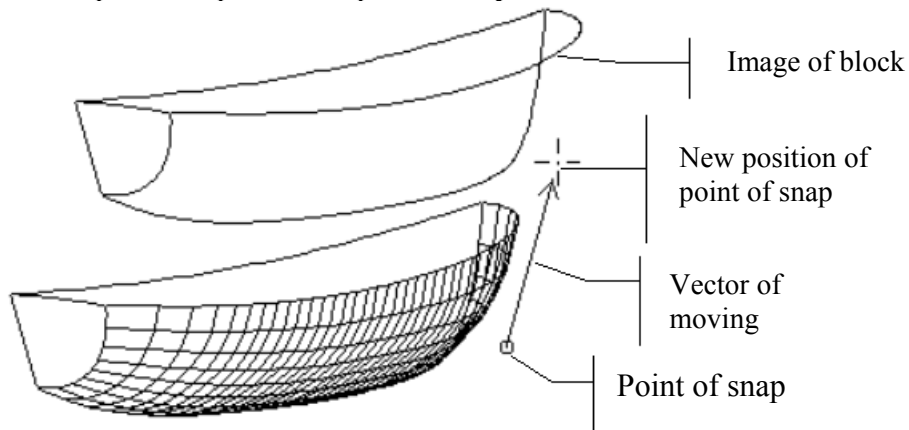
Remove

It uses for transferring elements from one block in another (in current). For transferring an element it is enough to specify of it. For convenience the current block for the period of performance of operation can be switched off. Then the transferred element will become invisible.

Move

Allows to move and turn blocks or groups of elements. Moving and turn are carried out in a working plane. Sequence of moving is following:

- indicate bloc or group of elements;
- indicate a snap point on enquire **“Basic point (Tab to rotate):”**;
- input a new point on enquire **“New position:”**.

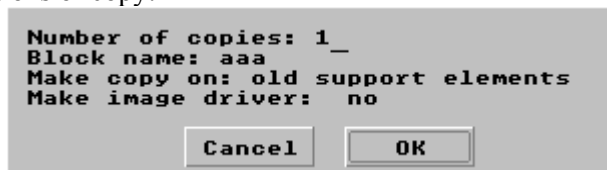


CopyMove

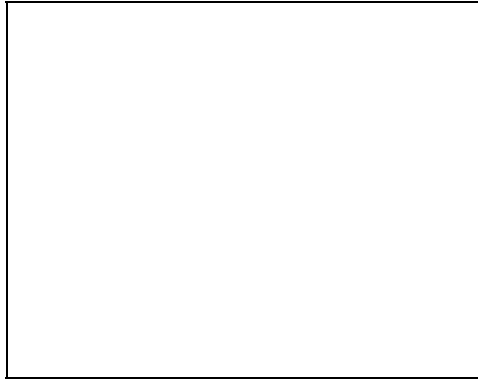
Allows to create copies of the block parallel carry of the block. It can be used for duplication of elements of the general arrangement, elements of a design

You should

- indicate block or group of elements;
- to set options of copy:



- **Number of copies: 1**
- **Block name: noname**
- **Make copy on: old support elements**
- **Make image driver: no**
- After the task of options by inquiry " Basic point: " to specify a point of snap of the block. You will see a copy of the block (a line of gray) on the screen
- You should input a new point of snap on enquire **“New position: ”**



CopyRot

This operation is similar previous. Difference will be, that instead of initial and final points of moving it is necessary to specify the center, initial and final corners of turn:

- **“Input center:”**
- **“Begin angle: ”**
- **“New angle: ”**

Cross

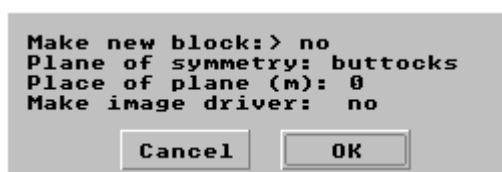
Allows to receive lines of crossing of all surfaces included cross in blocks (only for internal blocks)

Symm

Allows to execute symmetric image of elements of the block concerning the set plane of symmetry

You should:

- indicate block or group of elements;
- set parameters of operation in dialog window:



- **Make new block: yes**
- **Block name: noname**
- **Plane of symmetry: buttocks**
- **Place of plane (m): 0**
- **Make image driver: no**

VII. THE NEW PROJECT

INTRODUCTION

This part of the book covers the process creation of the file of a new project by an example of designing an elementary vessel hull. This chapter describes the main principles of dividing a vessel into separate parts for creation of pieces of a surface which in the total will make a valid mathematical model of a vessel surface. The chapter contains information about creating surfaces with using of different elements of the system with particular emphasis on the quality control of lines and surfaces.

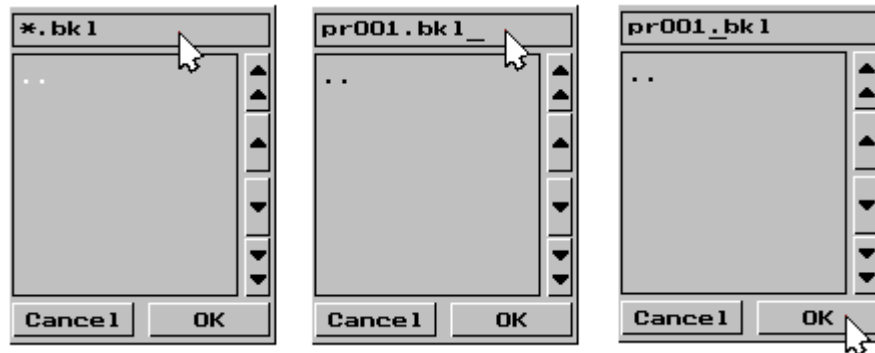
Using specific commands you will get familiar with all methods of activating commands, surveying a model, setting parameters and changing options in the dialogues.



VII. THE NEW PROJECT

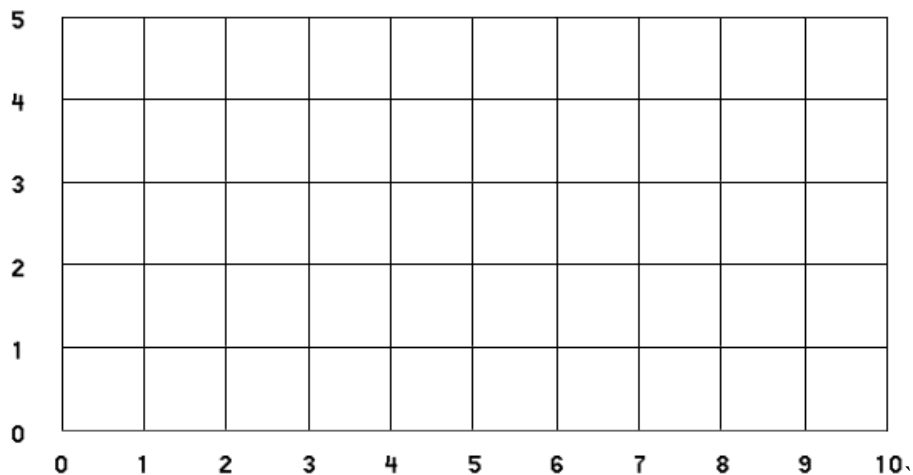
CREATING A NEW PROJECT

As in any system the commencement of operation always consists of opening the file of a new project and saving this file for a subsequent operation. You should start the system and enter a name of a file of a new project, using keys “← →”, “Del”, “BackSpace” and keyboard. Then press "Enter" or move a marker on button " OK "



SETTING THE GRID

By default the user coordinate system (UCS) coincides with an interior (mathematical) coordinate system and the lines drawing (LD) grid has the following dimensions: 100x50x50 meters with an interval of 10 meters between sections on each axis.



Let's set the LD grid as follows:

X-direction - the first area - 0,6, 0, 0,6, the second area - 0, 9,9, 0,9;

Y-direction - 0, 2, 0,5;

Z-direction - 0, 2,5, 0,5.

Let's set the X-axis direction from the right to the left.

You should use menu item **File/Grid** for setting up all required parameters

VII. THE NEW PROJECT

```
Grid of: ▶ absciss ( X ) :  
Direction of axis: left to right  
Numbering of sections: positive  
Position of zero section: 0  
Grid ( begin, end and step ) :  
1-d region: 0, 100, 10  
2-d region:  
Addition sections:  
1-d:
```

Cancel

OK

To change the direction on the X axis select the radio button “ **right to left** ”.

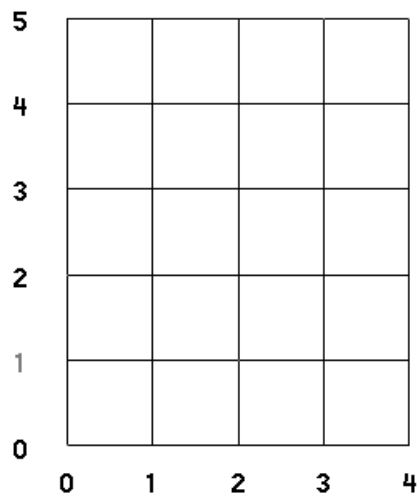
Press OK button, a grid with new dates will appear on the screen.

To change the view, select the **Front** button in the **View** toolbar.

The toolbars contain the functionally grouped commands. To enter a command, press the button of an appropriate toolbar.

There is a faster mode activating a series of commands. These are the so-called hotkeys.

In this case you can simply press **F5** to change the view for **Front**.



SAVING A NEW PROJECT

For saving the project, you should press **File/Save**. Now we have a file with created project and at any moment we can return to it for continuation work.

VIII. CREATION OF A VESSEL FRAME

INTRODUCTION

If you want to receive a model of a hull surface it is necessary to create border lines of separate surfaces. The design process in a system assumes designing of a hull with the help of several surfaces. In this chapter the emphasis will be put on creation of lines of the future surfaces of the vessel hull. There are certain rules on creation of lines to make a surface created on the basis of such lines well edited. Special attention further will be paid to such rules.



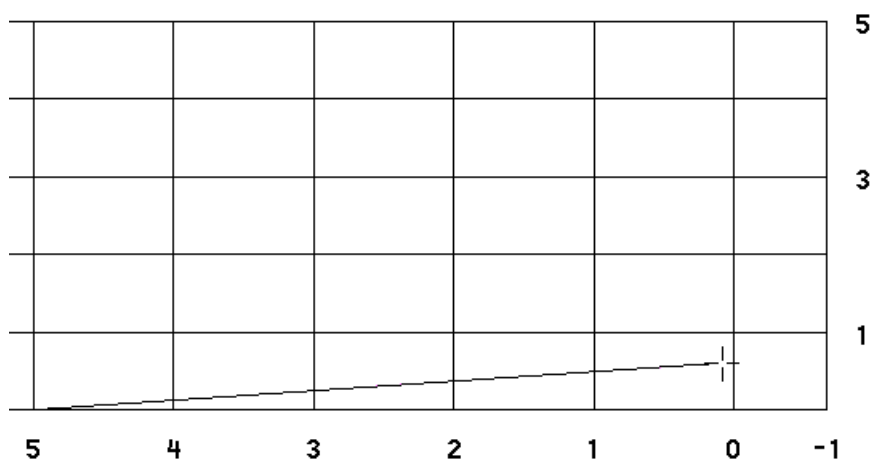
INPUT OF LINES

Let's draw a line of diametrical buttock from the 5th frame (point 4.5, 0, 0) to point 0.23, 0, 0.275.

Select the command **Line/Line**. You will see a following message in the request line: “**Input point: 0, 0, 0**”. If you press the keyboard key “←”, the marker will appear in the request line under the last digit.

Input point : 0, 0, 0

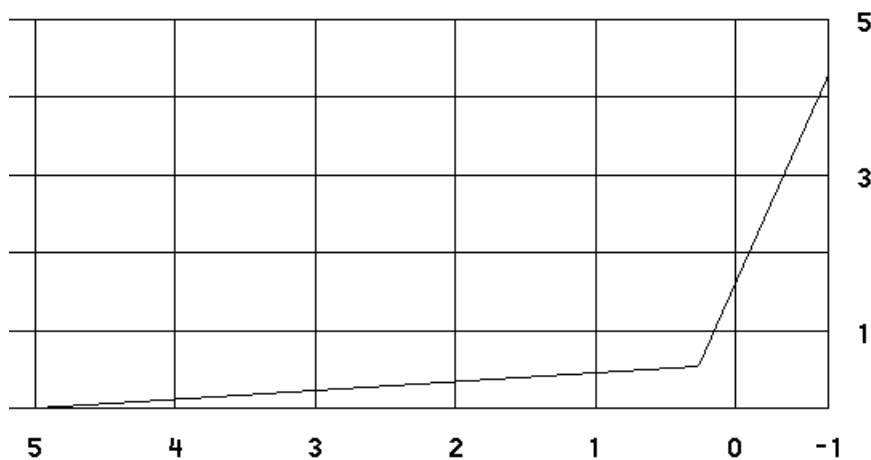
Instead of the first digit (coordinate X) you should enter 4.5 (or x5) and press **Enter**. Move the marker to the right in the area where it is necessary to introduce a second point of the line.



A so-called “rubber thread” of the prospective line will appear on the screen. In the prompt line enter the following digits: 0.1, 0, 0.3, and press **Enter**. The second point of the line will be introduced and its image of the current color will appear on the screen.

In the prompt line enter the following digits: x(-1), 0, y4+0.13 and press **Enter**.

The second line will appear on the screen.



Pay attention that by introducing the first point of coordinate X we have entered not a numerical value, but a numeral **x** and digit **5** meaning the number of a frame. This method can also be used with the other coordinates Y and Z. To obtain a negative value of a section it is

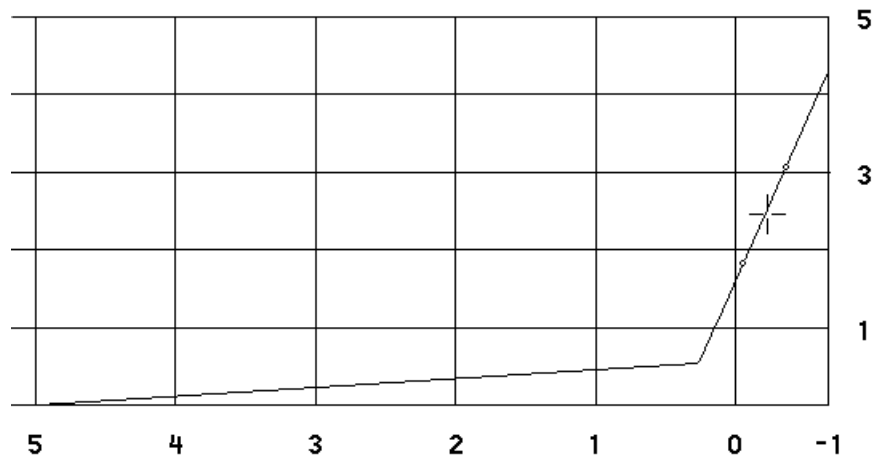
necessary to enclose the number of the section in brackets. Besides, it is possible to use arithmetical expressions, if the distance of coordinate from a frame, buttock or water-lines is known precisely. For example: $z4+0.13$ will refer to a coordinate along Z axis spaced from the 4th water-line over 130 millimeters. Use this method to input a second point of the following line, since the line condition mode is enabled.

Press the left mouse button or the **ESC** key. Now it is possible to introduce a new chain of lines. Press the left mouse button or the **ESC** key once again to disable the line creation mode.

Thus, two lines of the diametrical buttock of a new surface have been created. Further we shall consider modification of geometry of the introduced lines.

CHANGING THE NUMBER OF KNOTS

By default each introduced line has two knots. To check this click the mouse on the second introduced line.



Two control knots will be displayed on the line. Besides, we have started the element editing mode and can change its parameters. Generally, there must be at least 2 knots on the true lines.

To make changes select the following command: **Edit/Knot**.

The dialogue box for editing the number of units will appear on the screen.

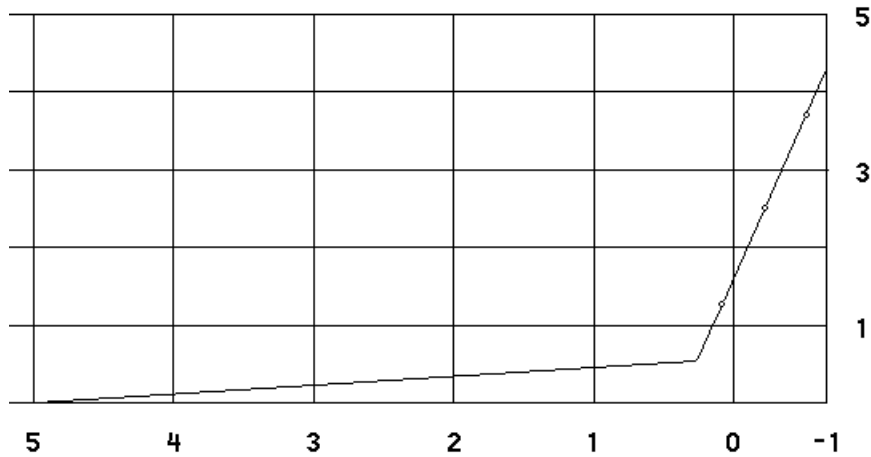
Knot to: add – choose necessary operation:

add

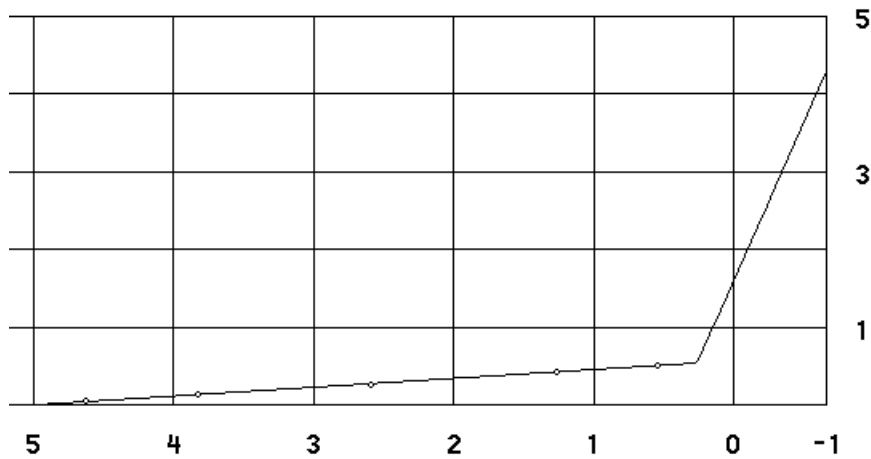
delete,

make even parametrisation

To change the number of control knots it is necessary to select the option **Make even parametrization**. In the window **New number of knots** it is necessary to enter digit 5. The end and the beginning of the line are also taken into account at tallying the control knots for the line. The line is more qualitatively defined by the uniform parameterization of the knots allocation.



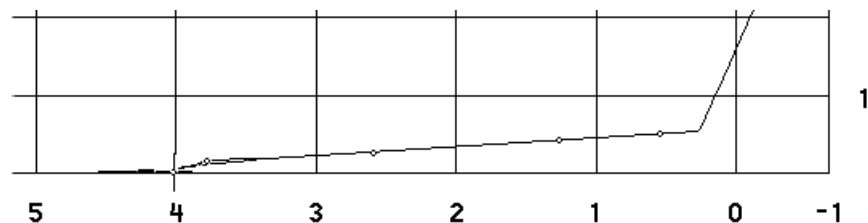
Let's change the number of control knots for the first line too. For this purpose we shall select a line for editing and enter number 7 in the following window: **New number of knots**.



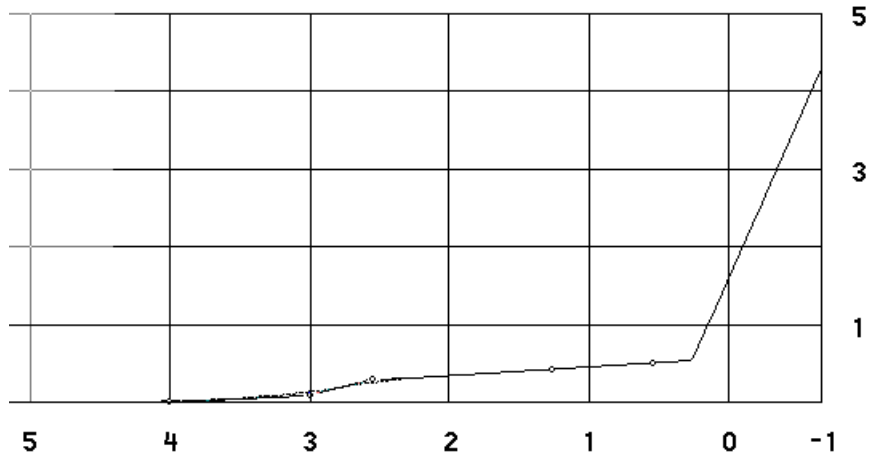
The number of control knots can be changed at any moment.

APPOLOCATION OF CONTROL KNOTS

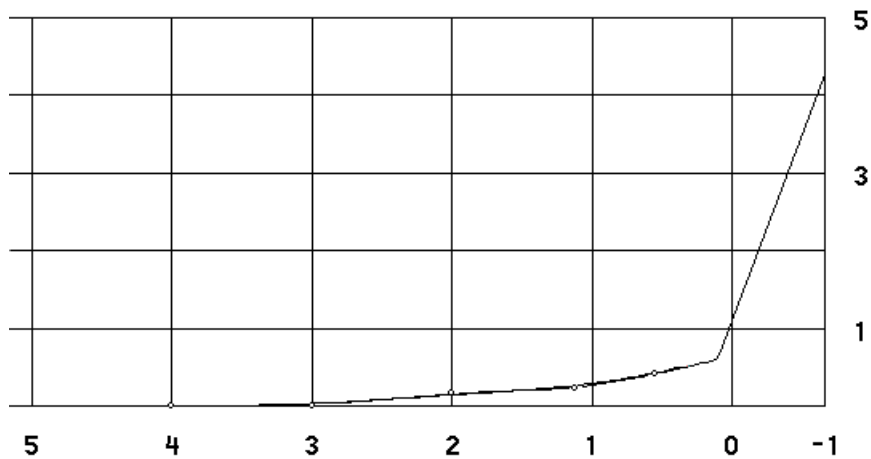
After that it is necessary to uniformly place the knots. For this purpose use the object snap to the grid. Further it will be required for creation of a surface on these lines. After the new number of control knots has been specified the line is in the editing mode. Without quitting the editing mode press **Grid** button in the Filter menu. The button will remain "depressed". After that with the help of the mouse select the first control knot of the line near the 5th frame and move it on intersection of the 4th frame grid.



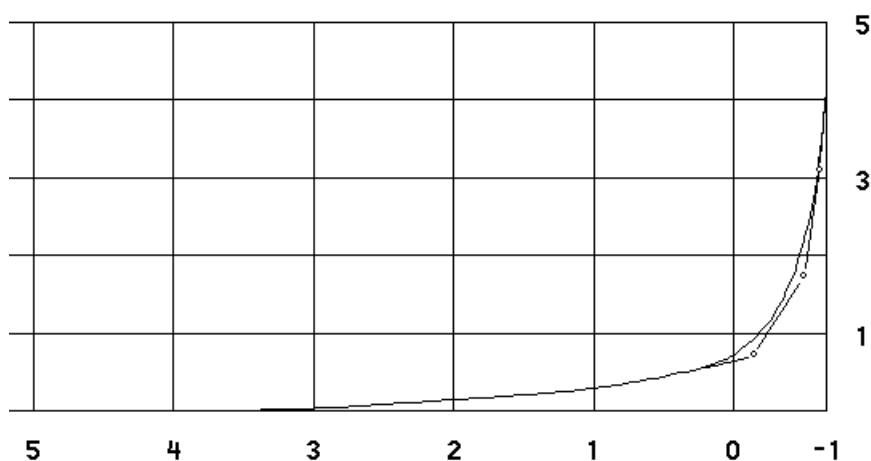
The second knot on coordinate X is compatible with the 3rd frame. For this purpose we shall use the **GridX** object snap. Without quitting the editing mode press the **GridX** button in the **Filter** menu and place the control knot so as to make the line from the 5th to the 3rd frame look smoothly.



The third knot can be matched with the frame grid of the project too. And the fourth and the fifth knots must be placed between the 2nd and 0 frames, beforehand having the **GridX** object snap rejected. For this purpose press the button. The button will be released from the “depressed” condition. The line will assume the following image.



Arrange the knots on the second line approximately in the way it is shown in the figure below.

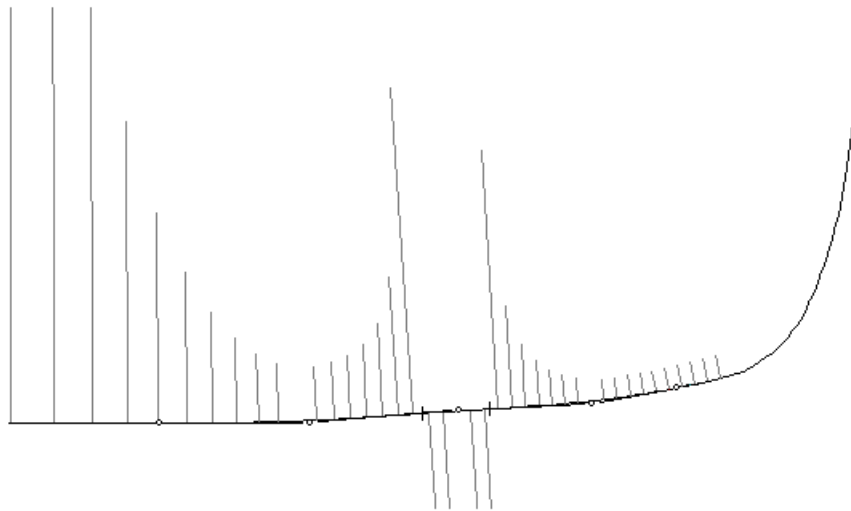


Thus, we have approximately placed the control knots on two lines. In a similar way the control knots will be placed on any lines.

MONITORING THE CURVATURE OF A LINE

At the following stage the grid image is not necessary and we will disable its displaying. For this purpose press the **Grid** button in the **Level** menu. The system implements a means of monitoring the curvature and inflection of the lines and surfaces. We shall start the procedure of final smoothing from the side of the second line, since the first line will be smoothly connect to the second one.

To display the radiuses of the line curvature it is necessary to execute the following command: **Edit/Options**. Enable the **Draw** option in the dialogue box that will appear. The **Scale curvity radius by** window contains the scale coefficient 100. This coefficient is inverse, it decreases the true value of curvature in the given point of the curve. It is necessary to enable the **Draw** option in the group of options **Bending** to display the inflection points. After enabling these options the following image will appear on the screen.



On the first line two points of inflection are visible. Within the area from the first point to the second point of inflection of a curvature line the lines are directed to the side opposite to remaining lines of curvature.

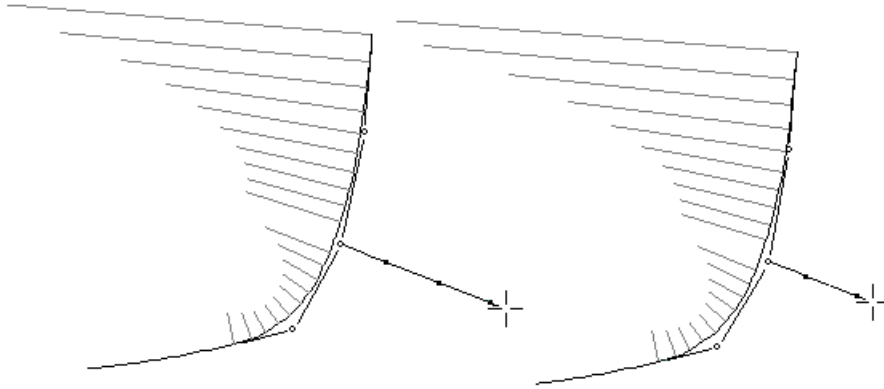


The second line has no points of inflection. Start editing this line.

For more exact editing of control knots select the following command: **Marker/Options**.

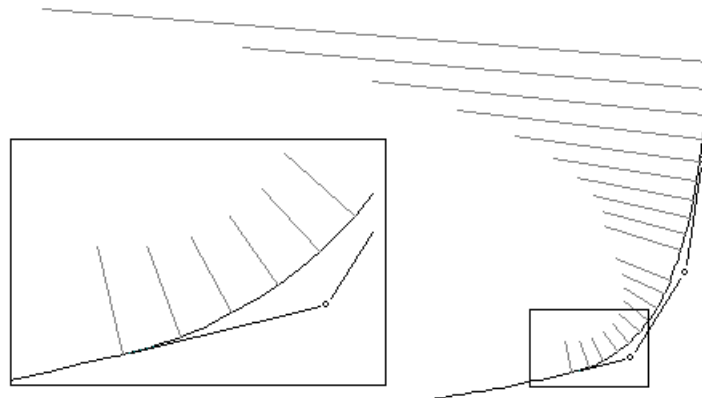
In the dialogue it is necessary to enable the **Scale** option. In the window **Scale value** enter the value of 5 mm. It can be set to the minimal value of 0.01 mm. This command and the mode for

changing the parameters can be invoked without quitting the editing mode.



While editing the second control knot a rubber thread with nodes will appear. The distance between the nodes will correspond to 5 millimeters in this case. Pay attention on how the radiuses of curvature of the line vary. In the process of smoothing the parameter defining the amount of movement needs to be decreased.

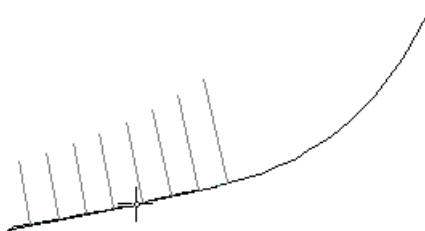
While smoothing this line it is necessary to pay attention on a lower control knot, i.e. on the control broken line. The direction of this broken line should coincide with a desirable direction of the first line in a point of their joining. As a result of smoothing the line should assume the following approximate image.



JOINING LINES

To avoid joining the lines by eye the system includes a command which allows butting one line to another. Thus the knots of the jointed line are automatically readjusted to make joining of two lines smooth.

Before executing the command for joining, it is necessary to select the first control knot on the first line which is located at the junction point with the second line.

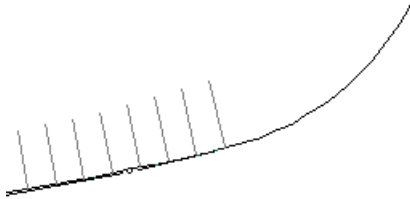


After that select the following command: **Edit/Joint**. It is admissible that while moving the

mouse the line may be strongly deformed.



In the dialogue you can select joining of lines not only by a surface, but only in the selected projection. After confirming the method of joining, the first line will be smoothly joined with the second one.



After joining avoid moving or otherwise affecting the first control knot. Besides, it is necessary to execute the **Edit/Options** command and enable the **Do not modify** option in the group of options **Modify connect elements** of the dialogue box. It will allow avoid changing the adjacent knots, while editing the control knot.

THE MODE OF ORTHOGONALITY

For the first control knot of the line for coordinate Z within area of the 4th frame it is possible to quickly set 0. For this purpose edit the third digit in the enquire line and enter 0. After that edit three control knots.

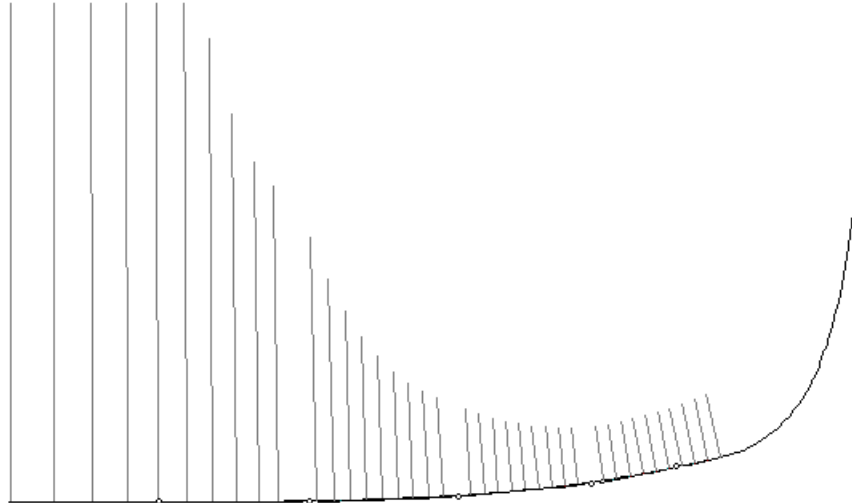
For their editing it is necessary to use the **Ort** mode to avoid shifting of the control knots in coordinate X. For this purpose press button in the **Marker** menu.

Any transition will be performed in a parallel way to the grid lines.



Using the **Scale** mode gradually change the magnitude of transition up to 0.1 millimeters while smoothing the line.

The second line should assume the following approximate image.



TOPOLOGICALLY CLOSED LOOP

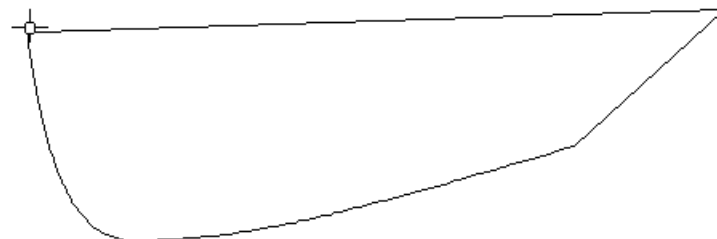
Two lines, which assumed the final image, have been initially joined topologically, while creating the chain. It occurs automatically while successively creating two lines. Further, while creating the lines, it is necessary to create lines with the snap from the **Filter** menu using the **Point** button. It will allow creating a surface on the basis of these lines. Each line in the loop of three or four lines should be linked to an adjacent line through the same point. If necessary, you may enable varying both lines while adjusting the common point.

Move to the isometric view. For this purpose execute the command from the **View/Isom** menu. Make sure that the **Point** snap mode is enabled. Select the command for creation of a line and move the cursor to the end of the line. Pay attention to the cursor image – the so-called “crosshairs”, which means that the object snap is enabled. In this case it is the **Point** snap.

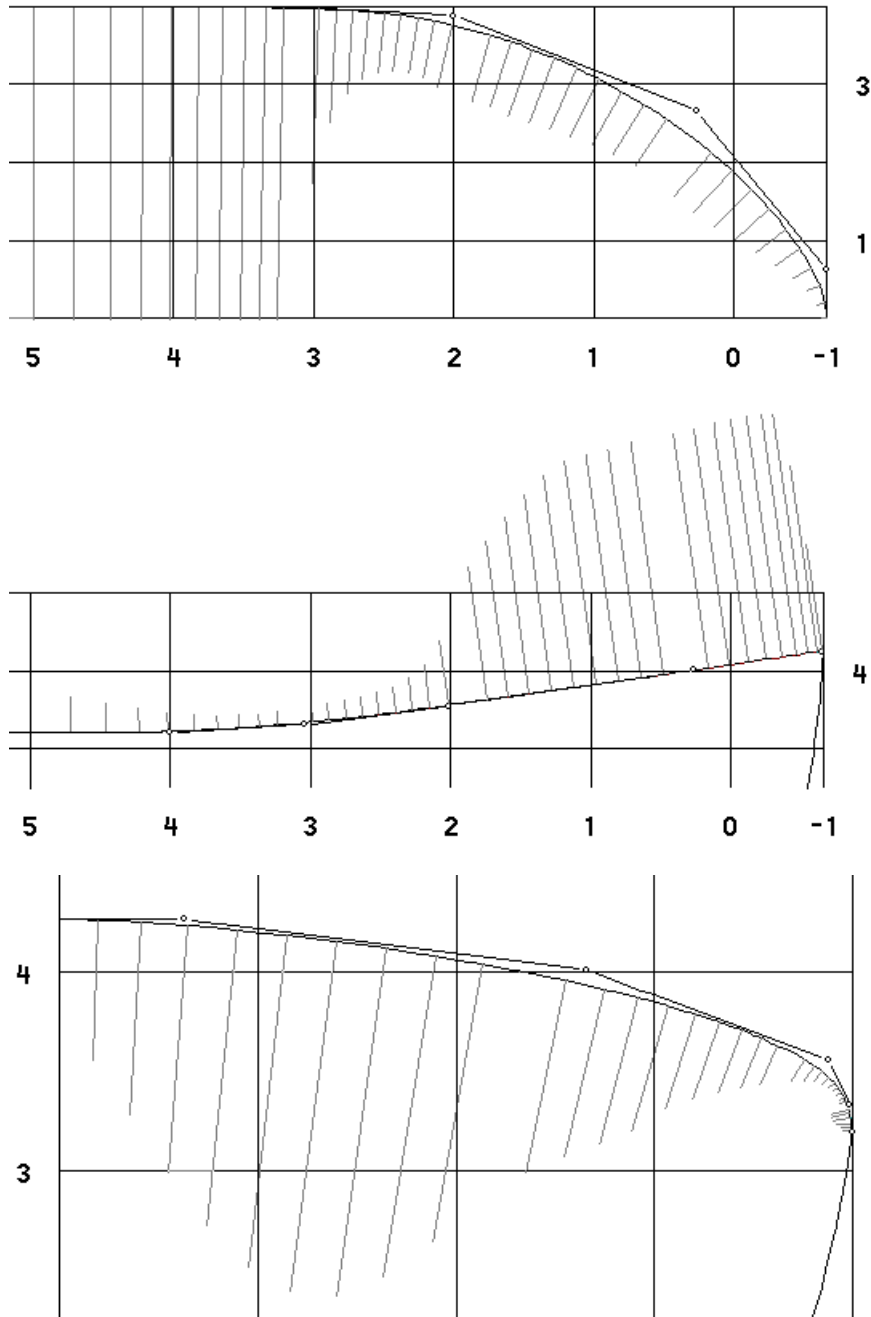
Click the left mouse button to perform the line point snap. A “rubber thread” of the prospective line will appear on the screen.



In the prompt line enter coordinates of the second point: 4.5, 2, 1.6. Then select the upper point of the second line of the diametrical buttock without disabling the point snap.



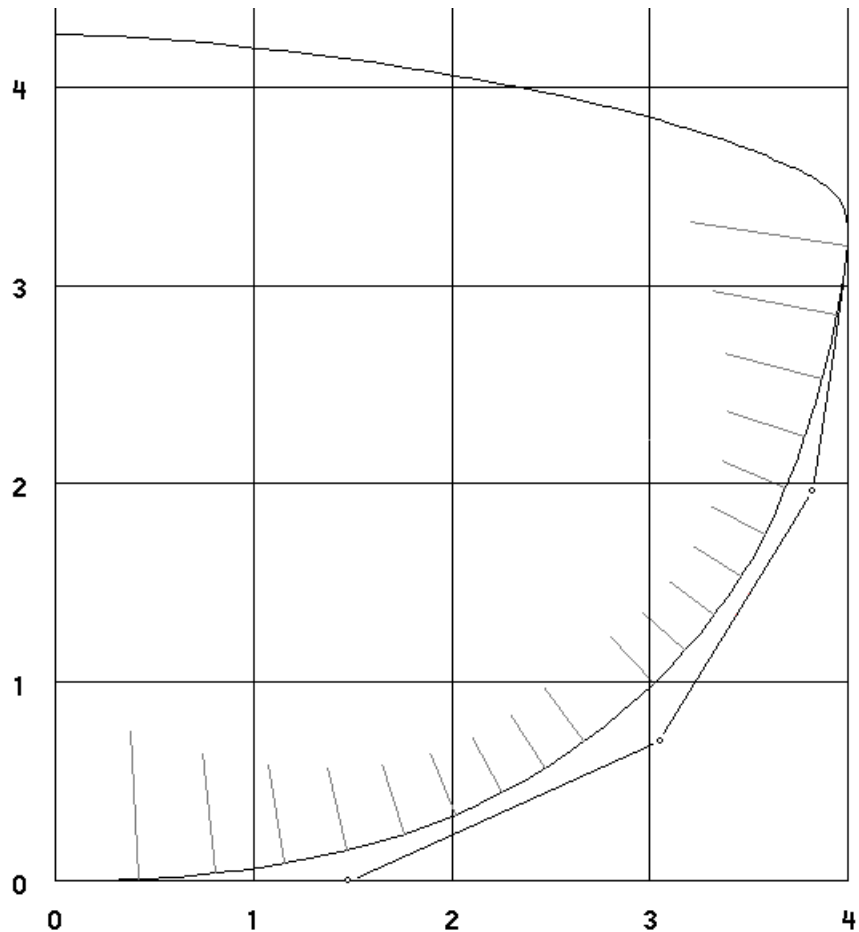
Arrange the knots on the sheer line at side and smooth it as shown in the figure below. For coordinate **Y** of the control knot on the 4th frame enter 2 and 1.6 for coordinate **Z**.



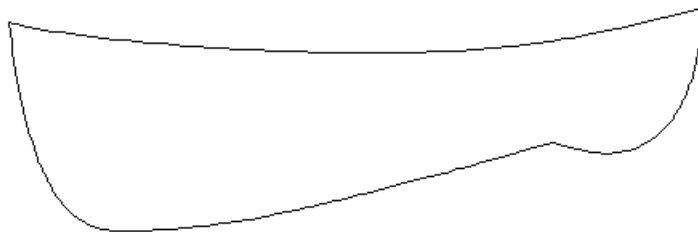
THE MODE OF ORTHOGONALITY

Pay attention, that the first knot on the 1st frame in the **Plan** and **Front** view should have the coordinate along X and Z which is equal to a maximum length and height of the line. Such mode is called the orthogonal mode. For this purpose there is the **Edit/OrtLin** command. As for the **Joint** command it is necessary to select the given control knot before executing the command.

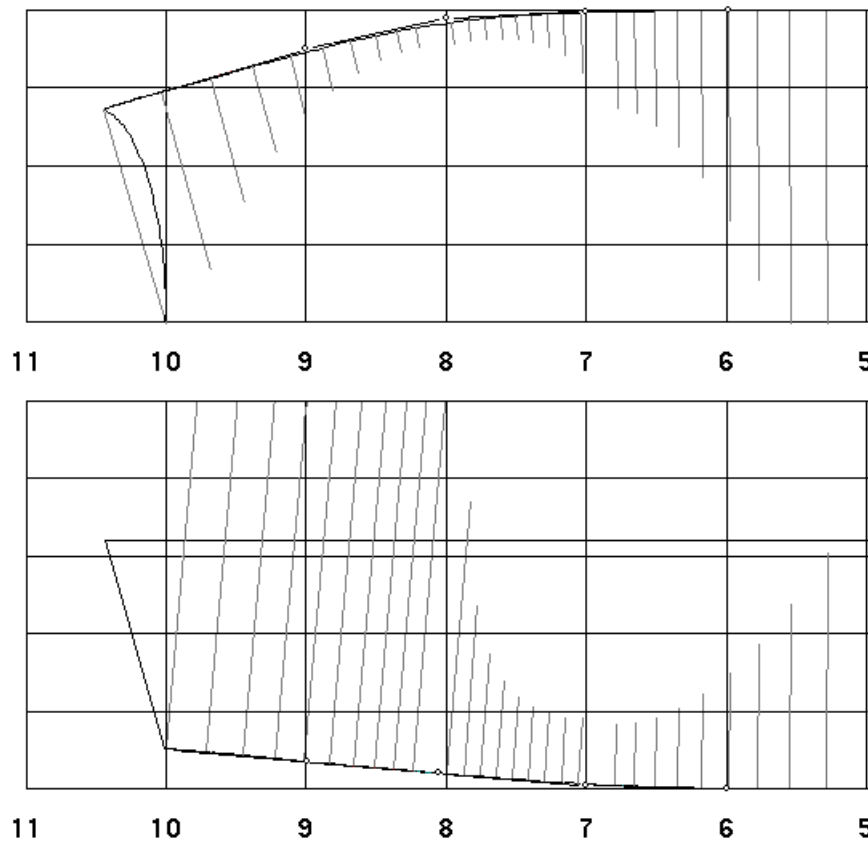
Arrange the knots on the line of the 5th frame and smooth it as shown in the figure below.



Thus, the frame of the vessel bow surface has been created.

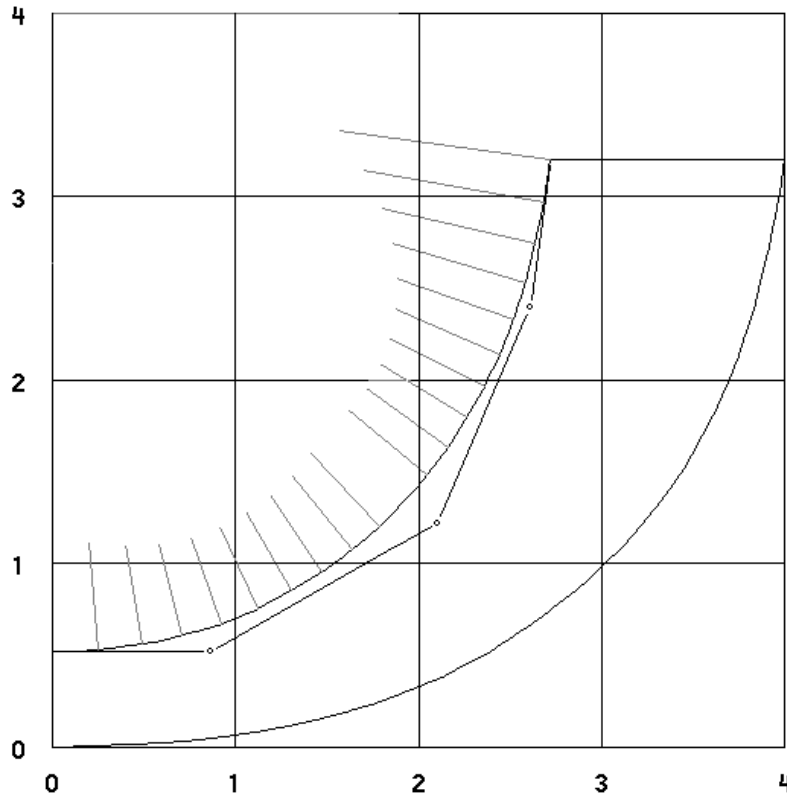


The, using the object snap to the point, complete creating the three lines of the stern surface of the vessel with the number of knots equal to 6 for the sheer line at side and diametrical buttock line. For the transom line, as well as for the line of the 5th frame the number of the knots should be equal to 5. For coordinates of the transom line points enter the following digits: 9, 0, 1.6 and 9.4, 1.36, 1.6.



The control knots of the lines, as for the frame lines of the stern surface, it is necessary to arrange on a single coordinate **X**. The sheer line at side and the diametrical buttock line should be joined with appropriate lines of the frame of the bow line. After that avoid editing these control knots. (Coordinate **Z** of all the control knots of the sheer line at side 1.6., i.e. the sheer line at side in the stern is horizontal).

The transom line in the **Front** view should look as follows:



In the **Side** view the transom line will be straight. To arrange the control knots in the **Side** view on a straight line, it is necessary to execute the **Edit/Strght**. The editing mode should be enabled for line.

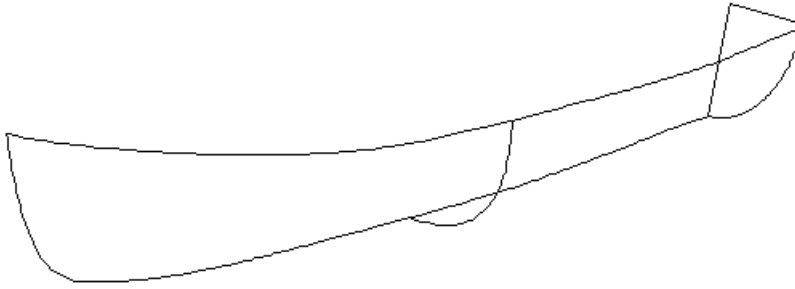
The dialogue for editing the position of control knots shall appear on the screen.



For saving the disposition of control knots in the **Front** projection, it is necessary to select the appropriate radio button in the **Saved Projection** window. As a result of these operations the line in the **Side** view will be straight.

The, using the object snap to the point, complete creating the two transom lines without changing the number of control knots, since the transom surface will be flat and arrangement of the control knots inconsiderable. The coordinate of the common point of these lines must be as follows: 9.4, 0, 1.6.

Thus, we have completed creating the frame lines of the vessel hull surfaces.



THE PROJECT TREE

Before you start creating surfaces on the basis of created lines, it is necessary to start creating the project structure. For this purpose the system provides for the project tree. To display the elements not only by the types, but also by the functionality it will be necessary, while including new elements to the project, to add the blocks to the project tree, which will include the newly created elements.

In the given case it was necessary to create a new block at once, for example, having the name **Lines** to locate there all lines of the vessel frame. It is not late to do it now, moreover, you will learn how to transfer elements from one block to another.

In order to activate the project tree, it is necessary to execute the **Block/ Tree**.



To create the block **lines** it is necessary to select the **Add Block**. In this case the new block will be an affiliate block for the current one.

A new block you can make a current block if you answer Yes on system enquires **“Do you want this block be current (y)?”**

There is a possibility of shifting the elements from one block to another. For this purpose it is necessary to make the block, to which you want to transfer the elements, a current one.

You can move blocks on the tree using **“Alt + ←”, “Alt + →”, “Alt + ↑”, “Alt + ↓”**

Thus, the structure of the project is created as required. It is necessary to create the necessary blocks at once, make them current and create elements inside them. In the given example it has been clearly shown how you can move the elements from one block to another.

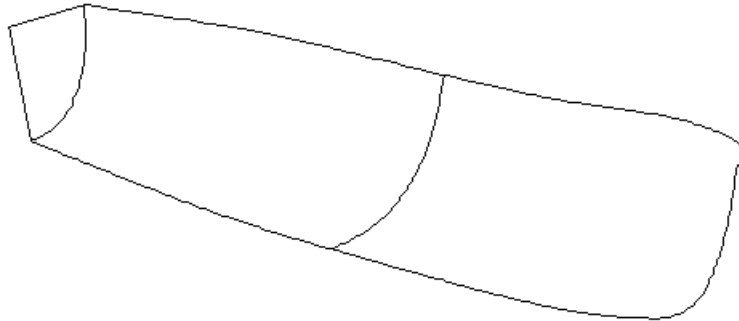
In this chapter it has been shown how you can create a vessel frame, while monitoring the line curvature, and arrange the elements in the project tree.

IX.CREATING VESSEL SURFACES

INTRODUCTION

After a vessel framework has been created, you can proceed to creating vessel surfaces.

After this the model on the screen will look as follows.



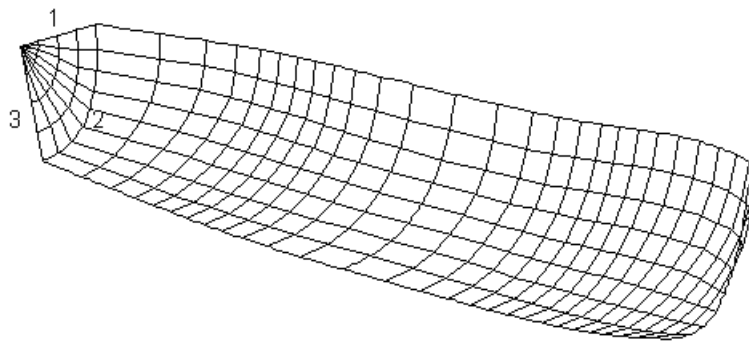
This chapter covers the methods of creating vessel surfaces including the deck surfaces and how you can cut them off.

CONFIGURING SURFACES

Select the command in the menu **Surf/ Make**.

The following message will appear in the prompt: “**Select lines and press Enter:**”. Specify any four lines of the bow surface. After being selected the line changes its color for red. It means that the line is selected for creating a surface and control is taken to make the lines to be topologically linked between each other. Press Enter. The surface will be created and displayed on the screen as a rectangular grid. In a similar way configure a stern surface.

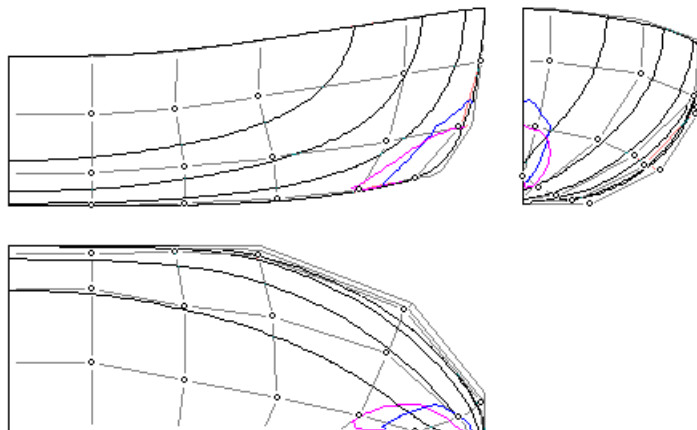
For a transom surface created on the basis of the three lines the sequence of selecting the lines is critical. The lines which form a sharper angle (lines 1 and 3) shall be specified in the following order:



SURFACE BEND

AREA CONTROL

While editing the system grid surfaces it is possible to control the surface bending areas. For this purpose execute the following command: **Edit/Options**. Enable option **Draw** in the group of options **Bending** to display the surface bending areas. Enable displaying of the surface bends in all three projections **Front**, **Side**, **Plan**. At this stage disable the option of displaying curvature of the surface sections: **Centers**. Enable displaying of sections and disable displaying of surfaces in the toolbar **Levels**. Select a bow surface to be edited and look at the surfaces from each three projections.



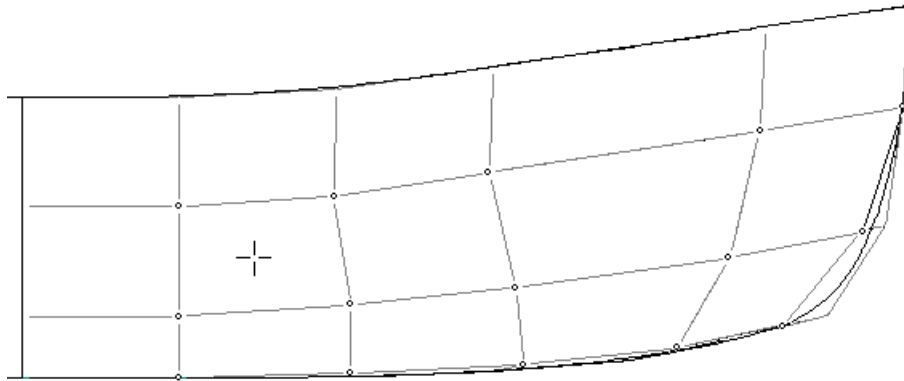
The bending areas of frames are displayed in blue color and the bending areas of buttocks – in red (not shown in this example), and the bending areas of waterlines – in purple. These areas segregate the surface areas containing a surface concavity which differs from the common

curvature of the total surface. By the sections you can see that they change the curvature pattern in the intersection point of the bending area boundaries. Thus, you can precisely determine all bending areas on the surface. Disable the sections for the first stage of the bending area surface editing.

ARRANGEMENT OF CONTROL KNOTS

Enable view **Side** and point the cursor to the place where the bow surface will be located.

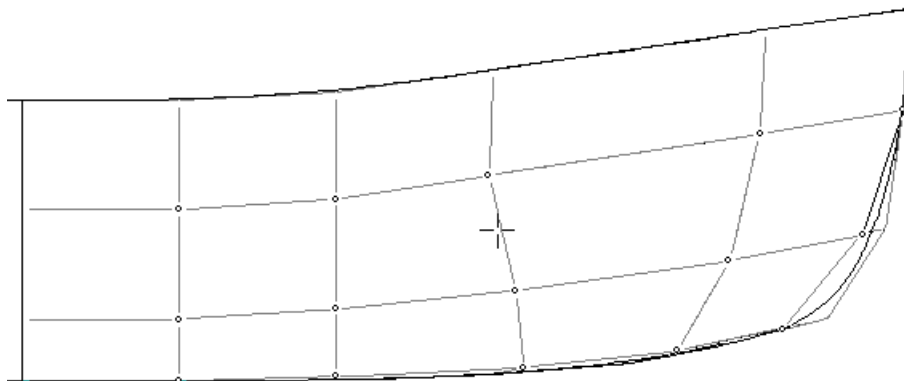
Control knots and control broken surfaces will appear on the screen.



The task of the first stage is to properly arrange the surface knots in all three projections.

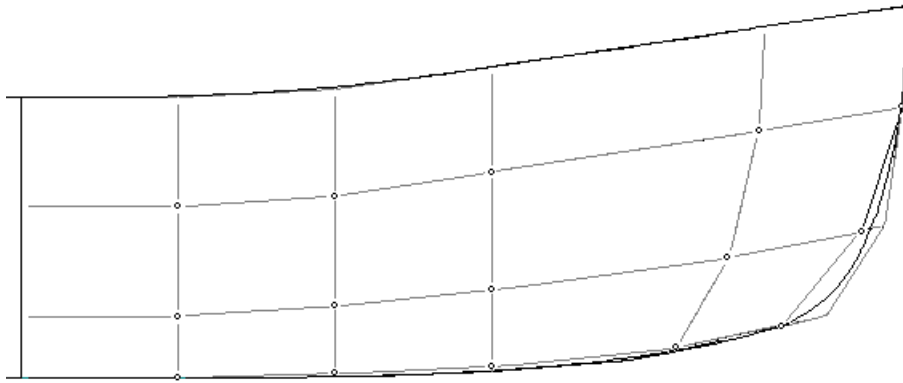
First, ensure the control knots of the second and third vertical row from the 5th frame are on the same line. It is possible since the knots on the opposite lines in coordinate **X** coincide.

Use the **Modify/Edit/Strght** command, which was used to make the transom line in view **Side** straight. In the dialogue box select **Front** for the saved projection.

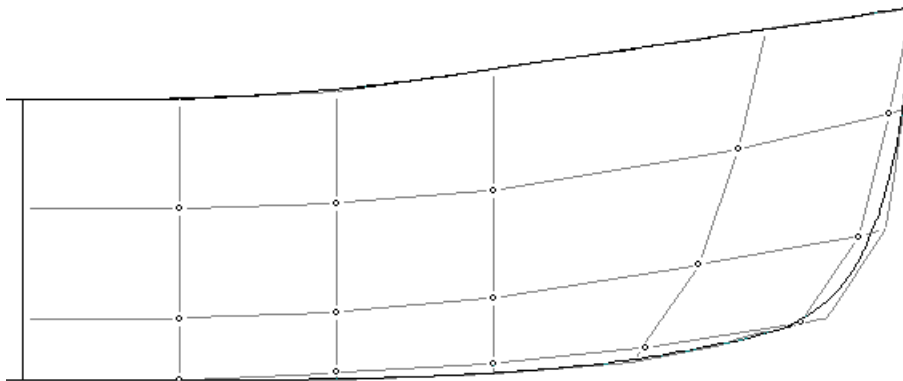


Point the cursor to the control broken line of the second row.

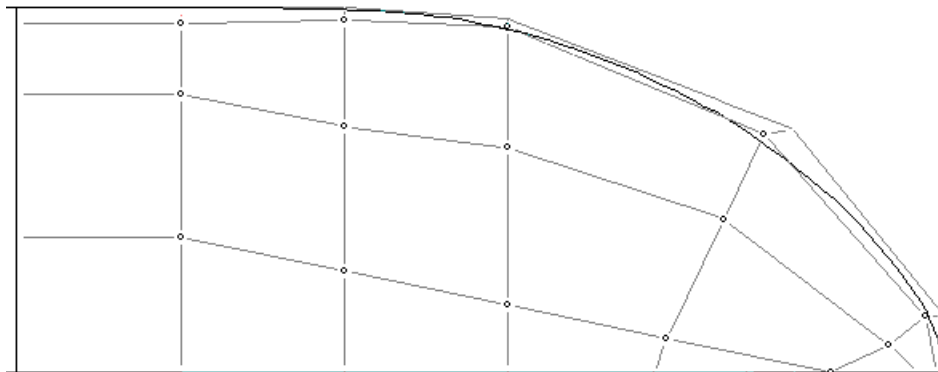
The second row of the control knots will become vertical. Point the cursor to the control broken line of the third row.



Then arrange all the control knots. Use mode **Ort** for the second and third rows of control knots. Do not edit the knots of the first row.

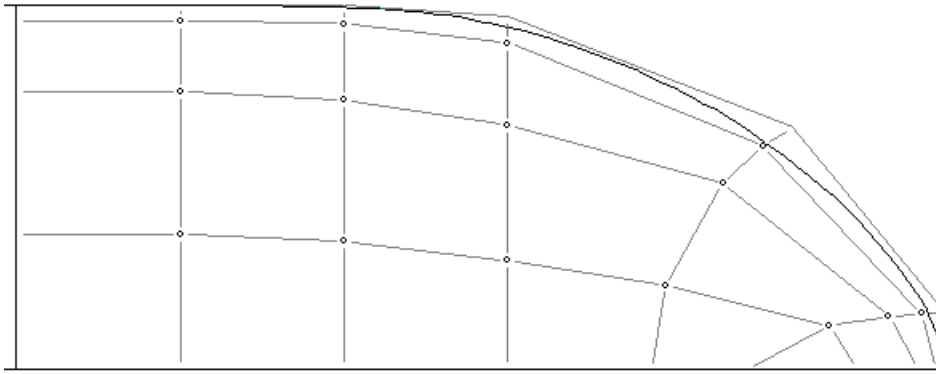


Enable view **Plan**. Control knots and control broken surfaces will appear on the screen in the given view.

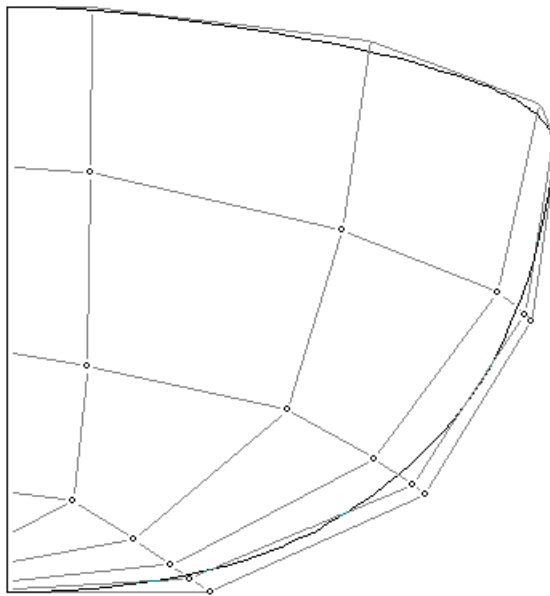


Then arrange all the control knots as shown in the Figure below. Do not edit the knots of the first row. Use mode **Ort** for the second and third rows of control knots. This will make it possible not to move the control knots in coordinate **X** and, therefore, not to “disturb” arrangement of the control knots in **Side** view. Other control knots will be arranged with the **Ort** mode disabled.

Try to ensure the broken line of the control knots would gradually change over from the horizontal state and turn to the horizontal line in the bow area. The control knots shall be approximately arranged as shown in the Figure below.



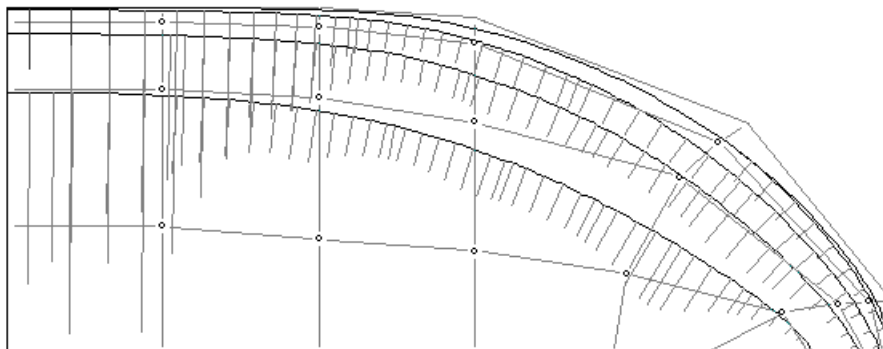
Arrange all the control knots in the **Front** projection as shown in the Figure below. In this case movement of the control knots will be inconsiderable since in the previous projections the control knots were arranged approximately in all three coordinates.



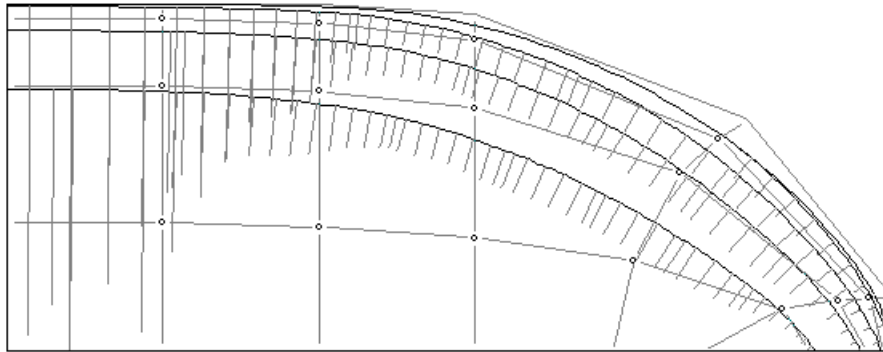
Thus, you can proceed to final straightening of the surface. For this purpose enable control over the surface bending areas. During the final straightening set the marker for short movements and, if necessary, eliminate local dents on the surface.

SURFACE CURVATURE CONTROL

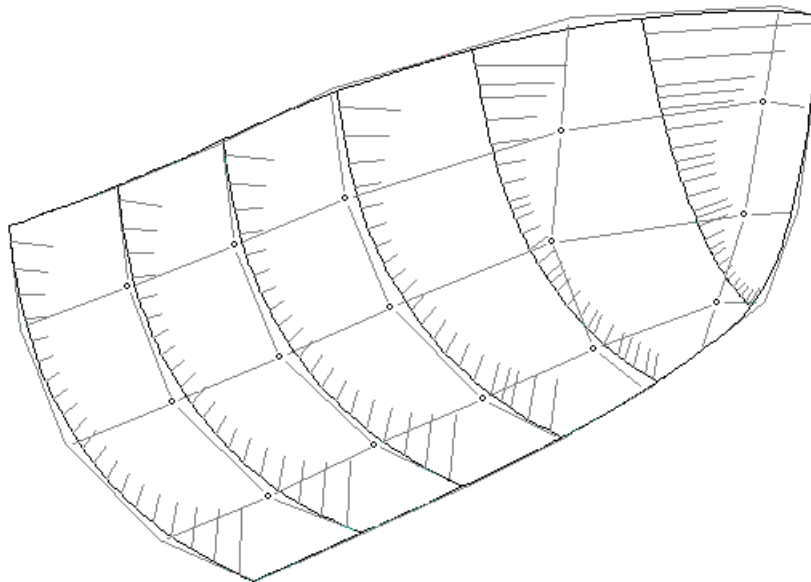
To view curvature of the surface enable option **Centers** displaying curvature of the surface sections. The same option is enabled while editing the lines. For the surface it is necessary that each section would have a gradual changeover of the surface curvature radii. Enable view **Plan**. Select a surface editing mode.



Follow the same rules as for arrangement of control knots on the surface using the short movement mode. In the final stage the surface will assume the following approximate view.



The system allows editing a surface in isometric view, which is most convenient when editing the surface curvature of frames.



To ensure the control knots would vary only in the Front plane it is necessary to adjust some parameters.

CONFIGURING THE DECK SURFACE

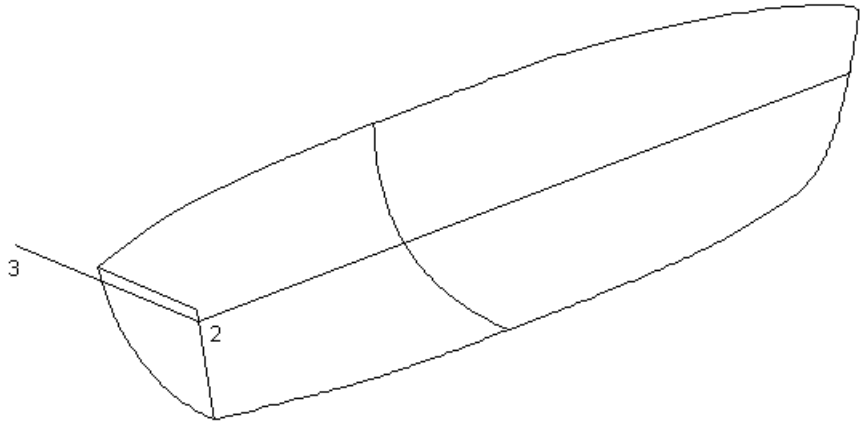
Before proceeding to configuring create a new block **Deck** in the project tree where the newly created lined will be located.

The system incorporates drivers as a convenient tool for creating a deck. The drivers can also be used for configuring surfaces of revolution, making radius conjugation of lines and forming other elements that are automatically maintained during correction of the source elements of formation. Study the basic operating principles of the drivers by example of the elongation driver.

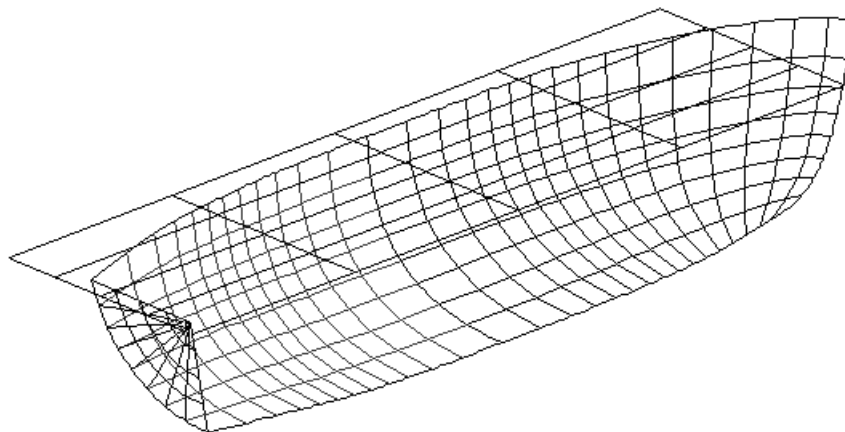
First, look at the source elements: the directional line and the line which will be elongated along the generating line thus forming a surface. While creating the elements use the topological snaps **Point** and **Line**. This will allow while correcting these elements to automatically change all surfaces of the driver, since there might be several generating lines.

IX. CREATING VESSEL SURFACES

To begin with it is necessary to create a line from point 1 to point 2 with the snap by Line and coordinate Z. Enter 1,45 for coordinate Z. To create the second line (from point 2 to point 3) substitute 0 with 2,5 for coordinate Y in the prompt line.



To create the elongation driver select the command from the menu **Driver/Extrude**. In the dialogue box confirm the type of creation: **By direct line**. For request “**Select director line**” specify a line in CL. An arrow will appear which will show the surface elongation direction. For request “**Select source line and pres Enter**” specify the second created line and press **Enter**. Two more lines will appear on the screen. With the help of the **Levels** toolbar enable displaying of the surfaces.



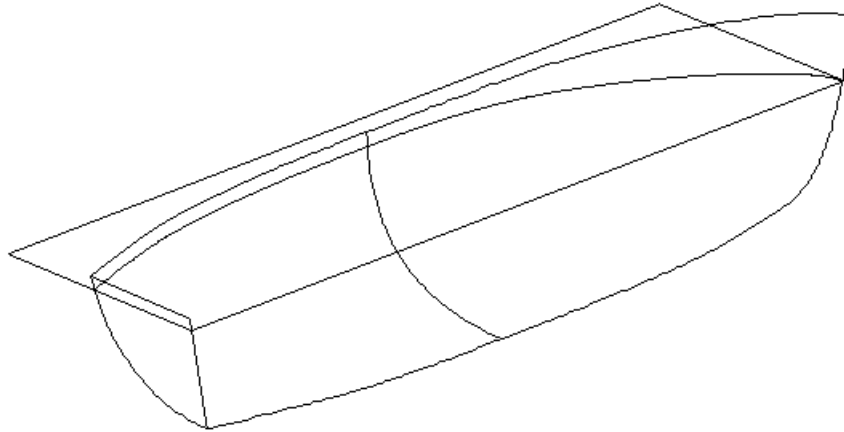
An image of the surface will appear which boundaries will be the two lines that you have created and two lines automatically created by the system. This is the driver surface.

A difference between the driver surface and a usual one is that you can edit only the source elements. If necessary you can break the driver and turn it into a usual surface.

CREATING INTERSECTION OF SURFACES

Now you need to create an abutment line of the deck and the vessel plating. For this purpose the system implements the option of finding intersections of surfaces. Before you start finding the intersection lines disable the surface displaying option in the **Level** menu.

To create intersection lines select the command from the menu **Line/Cross**. For request “**Select secant surface or driver:**” specify the deck surface. For request “**Select selected surface or driver:**” specify the bow surface and then the stern surface. While specifying the surfaces the intersection lines will be automatically created.

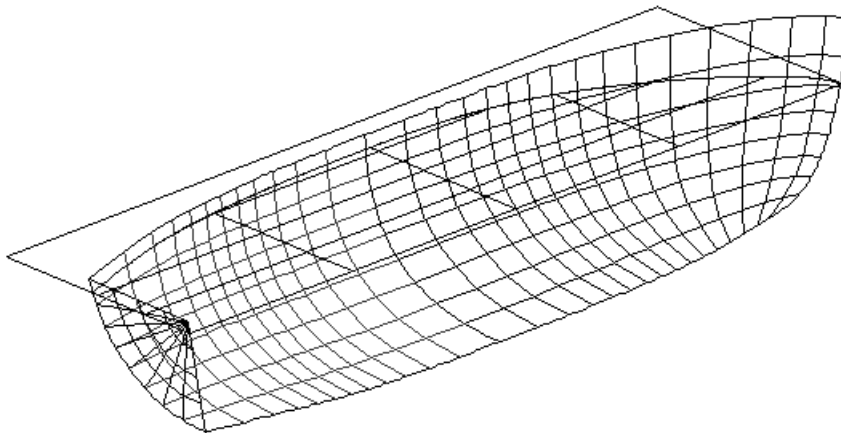


CUT OUTS IN THE LINES AND SURFACES

After you have completed all the required formations cut out unnecessary portions of the lines and surface. For this purpose there are special commands for cutting out the lines and surfaces. You can also enable or disable displaying of the cut out portions of the lines and surfaces. In the toolbar **Level** enable displaying of cutouts.

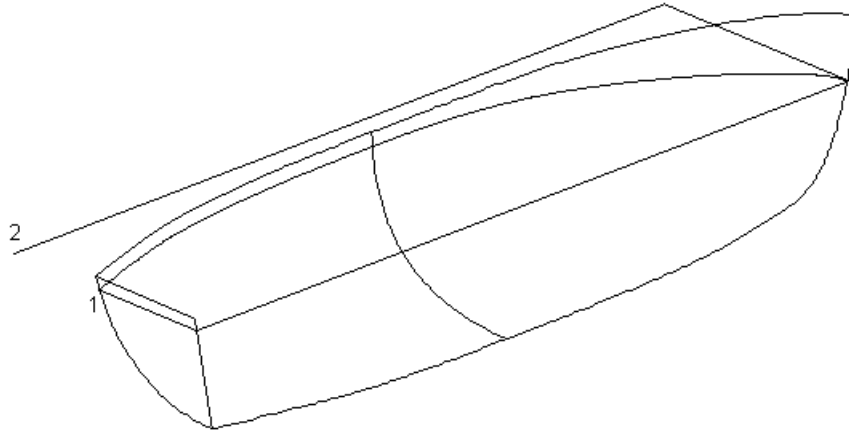
To make a cutout in the surface select the command from the menu **Surface/Cut**. For request “**Select lines and press Enter:**” specify the contour consisting of intersection lines of the surfaces and lines in CL.

The surface inscribed in the specified contour will disappear.



In fact, the cut out portion of the surface still exists, but it is not displayed and thus will not be present in the resulting documents. If it is necessary to display the data on geometry of the cut out portion of the surface or line simply disable displaying of the cutouts in the **Level** menu.

Also disable displaying of surfaces for creation of cutouts and select the command in the menu **Line/Cut**. For request “**Select begin point:**” specify point 1 as shown in the Figure below and for request “**Select end point:**” specify point 2.



The points should be topologically present on the line. By this method you can cut out unnecessary portions of the lines.

Move generating lines of the driver to block **Tools**.

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APPENDIX A. Reference of commands

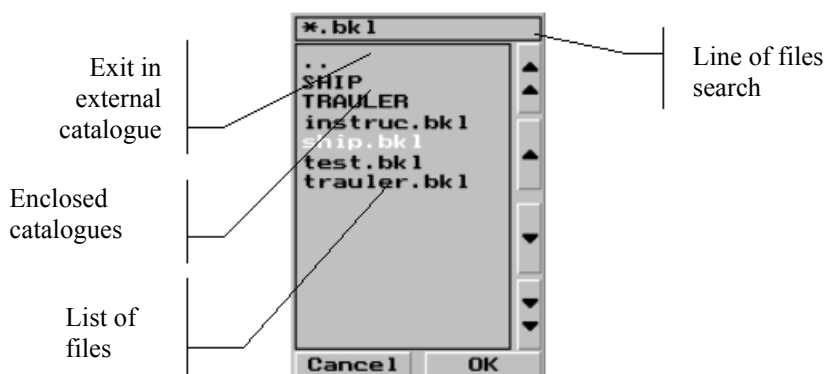
TEXT EDIT

Editing of text lines or numerical expressions in various dialogues is carried out by means of keys

“← ↑ ↓ →“, "Del" и "BackSpace".

WINDOW OF CHOICE OF FILES

The window of a choice of files is used for a choice of files from existing and for creation of new files



To create a new file (in item "File/Open" the bottom menu), pass to a line of search. For this purpose press a key "Tab" or move a marker to a line of search and press "Enter". Then, using keys “← →“, “Del” and “BackSpace”, enter a name of a new file, for example, "proekt01.bkl", and press "Enter" or button "Ok ". It will be system inquiry:



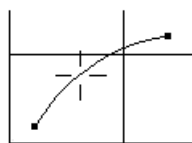
For confirmation press “Yes” ore “Enter”. For refusal press “Esc”.

SELECTION OF ELEMENTS

While operating the system you will have to select various elements for various operations, i.e. for editing, deletion, attribute variation and etc. This Section contains the information on how to select such elements.

ELEMENT INDICATION

To indicate an element, it is necessary to superpose the marker with the element (the element must get into the marker cross-hairs, see the Figure) and press "Space" key or enter its system name using the keyboard.



To indicate a surface, it is necessary to indicate the inner area of the surface ruler contour. To indicate a cutout, it is necessary to indicate the “cut out” part of the line or surface. To indicate a driver, it is necessary to indicate any of its constituent elements. Other elements are indicated in the same way.

While executing several commands, the indicated element becomes marked. A line is marked by changing its color to red. A surface is marked by changing color of its boundary contour.



A point is marked with the point indicator.

If after pointing with the marker the system finds several elements of the same type, it marks them in sequence and requests: "This element (y)?". The searching is effected among the visible elements falling within the current window in the following order: points, cutouts on lines, lines, dimensions, cutouts on surfaces, sheets, surfaces.

When indicating a driver element, depending on the executed command, either the indicated element or the driver will be found, or neither the element nor the driver will be found. For instance, let a line be a driver element. When executing the object tie-in to a line (“Filter/Line”), the line will be found. When indicating a line in the menu item “Edit”, a driver will be found (the driver elements are adjusted only if they are part of the driver). Finally, when indicating a line in the menu item “Delete” in the line deletion mode (“Delete/Line”), the line will not be found (the driver elements are deleted only if they are part of the driver).

INDICATION OF A CONTOUR OR A SET OF LINES

To indicate a set of lines it is necessary to sequentially indicate the included lines in the due order. The indicated lines will be marked in red color. To stop indicating the lines press "Enter". Pressing the "Esc" button will result in sequential canceling of indication and aborting of the command.

If the lines are to form a topologically linked chain or meet other requirements, the search will be performed only among the appropriate lines. If the specified line does not meet the requirements, a message will appear that will suggest repeating the indication.

To indicate a chain of lines it is necessary to indicate the initial and final point of the given chain. Then the system will automatically determine the closed chain. The lines included in the chain will be marked in red color. If more than one chain can be created between the initial and final points (with possible “forks”), on the system request indicate the next line of the chain, which is linked with the marked lines of the chain.

In some cases for indication of a lines chain you can subsequently indicate all of the lines (similar to indication of a lines set).

To indicate a contour it is necessary to indicate one of the lines of the given contour. Then the system will automatically determine the closed contour. The lines included in the contour will be marked in red color. If these lines can form more than one contour (with possible “forks”),

on the system request indicate the next line of the contour, which is linked with the marked lines.

INDICATION OF ELEMENTS

The system implements a possibility of indicating several elements at once (for example, to delete a group of lines, without selecting each one separately) by configuring the elements selection method. To define the elements indication method, use the following command: "Settings/Select". The system implements five methods for indicating the elements.

Element

This method of indicating separate elements is yet familiar to you.

Window

This method of indicating an element requires that the element would completely fall within a certain 3D window configured by the user on the system request. The window configuration procedure is similar to that for the current window.

Window Crossing

This method of indicating an element (as opposed to the previous one) requires that only a part of the element would fall within the configured window.

Tree

This method indicates a block in the dialogue box of a tree of blocks.

Block

This method of indicating a block requires that on the system request you should indicate any element of the given block.

INDICATION OF BLOCKS AND GROUPS OF ELEMENTS

Sometimes it is impossible to indicate all the required elements or blocks by a single indication. Therefore, some project commands implement a recurring request for indication. In this case the indicated elements or elements of the indicated blocks will be marked and the request on indication will be repeated. On indication of all the elements or blocks, press "Esc" on the system request for the "Tree" indication method and "Enter" for other indication methods.

On the recurring request you can change the indication method. Thus, a group may simultaneously include blocks and individual elements.

You should remember that selecting the same element twice will deselect this element. You can use this feature to deselect the unnecessary or unintentionally selected elements.

APPENDIX B. HOT BOXES

Input frequently used commands is possible from the keyboard by means of hot keys. For example if you press Alt and X it will be corresponded to press “File/Exit” in lower menu.

SYSTEM MENU

AltX, AltF4	File / Exit
AltF	File
AltE	Edit
F2	File / SaveAs
F3, CtrlO	File / Open
CtrlC	File / Config
CtrlE	File / Export
CtrlS	Сохранение проекта
CtrlF4	Заккрытие проекта
CtrlP	Filter / Point
CtrlL	Filter / Line
CtrlG	Filter / Grid
CtrlX	Filter / GridX
CtrlY	Filter / GridY
CtrlZ	Filter / GridZ
F4	Go
F5	Work Plane / Front
F6	Work Plane / Side
F7	Work Plane / Plan
F8	Work Plane / Screen
F9	Marker / Free
F10	Marker / Ort
F11	Marker / Scale
F12	Marker / Deep
AltF1	View / Front
AltF2	View / Side
AltF3	View / Plan
AltF4	View / Isometry
AltF5	Set / Isometry
AltW	Window / New
AltQ	Window / Prev
AltS	Window / Shift
AltA	Window / All
AltD	Window / Default
AltN	Window / Name
AltLeft	Window / Left
AltRight	Window / Right
AltUp	Window / Up
AltDown	Window / Down
AltB	Block / Current
AltT	Block / Tree
AltBackSpace	Undo

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ISOM

AltA	Edit of vector of turn
F1	“XY”
F2	“YZ”
F3	“XZ”

BLOCK TREE

CtrlEnter	Attrib
Tab	Change of mode of viewing

AltV	View
AltI, AltT	Information
AltR, AltN	Rename
AltP	Path
0 - 1, a - f	Change colors from 0 to 16



APPENDIX C. PROBLEMS

THE SYSTEM CANNOT BE STARTED

When you start the system it fails to load:

1. Check whether the protection key is inserted in the computer LPT port. It can be inserted improperly.
2. If a printer is connected to the key, disconnect it and/or turn it on or off.
3. Make sure the LPT port is in a good working condition.

Another case may be that the key driver has not been installed during installation of the system. Find the file “hldr32.exe” on the setup disk and install the key driver.

ELEMENT FAILURE

If the system does not draw an element, check:

1. Whether the corresponding display level is enabled (upper menu **Level**).
2. Whether the corresponding color and level in the block (for a line, surface and point) or a reference element (for cutouts) are enabled. If you are not sure about that, click **“Block/All on”**.
3. Start the operation of verifying the base (lower menu **“Tools/Check”**) with the option of checking the accessories (**“belongs”**).

If on execution of all these operations the element does not appear, it means that either the element is absent or this is a line or a surface which have been fully cut out, or these are two similar cutouts on the surface overlaid on each other.

ELEMENT CANNOT BE SPECIFIED

If you specify an element, but the system cannot find it, three situations are probable:

1. The element is not certainly appropriate for the given operation. For example, when deleting surfaces that are a part of the driver they cannot be specified since you can not delete a part of the driver. Check the type of this element with the help of upper menu **“Tools/Info”**.
2. The element is disabled but is not removed from the screen. Click button **“Go”**.
3. The system finds first another element with a higher priority. For example, the points are found before the lines and the lines are found before the surfaces.

RUBBER THREAD IS ABSENT

If you cannot see the rubber thread, it means that it either should not be drawn at all or the point from which it comes out (or the working plane) lies beyond the 3D window (for example, when rotating the block – the center of rotation). Configure a larger window.

THE RUBBER THREAD IS DRAWN INCORRECTLY

While working in perspective view always ensure the marker projection on the working surface is inside the overall window when introducing a point. Otherwise, the introduced point may be arbitrarily far from the overall window. Moreover, if you hit the marker above “horizon” of the working plane, position of the visible point is inverted.

CANNOT CREATE ...

Such messages may appear while executing most commands for creating new elements. Generally, such message indicates that a desired element cannot be created under the existing conditions. Some typical examples:

1. When introducing a superficial line, it may go out the surface limits or the surface may have strongly degenerate parametrization (for example, near the degenerate angle of trihedral surface due to correspondence of the near-border knots with the border knots).
2. When creating a tube, section or mating surface – the path line has coinciding knots.
3. When creating a mating surface on the radius – the line of radii has nonuniform X-direction parametrization, goes out to area $Y \leq 0$ and etc.
4. When creating a mating surface along the seam line – the seam line is “shorter” than the centre line, i.e. the surfaces which pass through some points of and perpendicular to the centre line, do not cross the seam line.

CANNOT MODIFY ...

When correcting the elements, moving and copying the blocks and other executing other commands, sometimes, the system cannot complete the procedure of modifying the project and the following message appears "**Cannot modify element ..**". Here are some possible causes, due to which the system cannot modify an element:

1. Due to variation of the source elements the element became degenerate or non-existing (for information on possible causes see the previous paragraph).
2. If a surface cannot be modified, it, generally, means that its angular points are displaced in relation to its true position on the border lines. Enable the option of verifying the base with checking the geometry by menu "**Tools/Check**".
5. If a line cannot be made straight in one of the projections saving its shape in the other projection (when executing the command "**Edit /Straight**"), it means that the third coordinates (i.e., depth in the third projection) of its beginning and final points coincide.

SYSTEM ERRORS

Other errors occur due to the system operation errors or due to its conflict with the environment (for example, with the operating system or drivers). This is a situation when the system behaves in a different way than what is described in the manual.

It is recommended to thoroughly process and register such errors in the documents as described above and send them to SEA SOLUTION support.

SEARCHING FOR THE BASE ERRORS

The system incorporates a regular method of searching for errors in the data base structure. With the help of this command "**Tools ► Check**" you can find and correct almost all possible errors in the structure of the project data base. In particular, you can partially restore the project, if it was corrupted at the file level. It is recommended to start this procedure at regular intervals for preventive purposes, for example, prior to and after implementing all considerable changes in the project.

Besides, the same procedure is used to restore some redundant data contained in the data base and intended for boosting its operation. That's why it starts automatically, for example, after



the operations on import of data from other formats. Besides, it is recommended to start it on obtaining a new system version or prior to working in the project which was created in other version of the system.

For this purpose enable the following options of the procedure in question (additionally to other settings):

- **check belongs: yes** – enable the option of checking whether each option belongs to correct “master”.
- **check references: yes** – enable the option of checking whether the references between the elements are correct.
- **check geometry: yes** - enable the option of checking whether the ends of the lines geometrically coincide with their end points and whether the hanging points are strictly on the reference line and etc.
- **Try to correct: no** - enable the error correction option.
- **Stop then error: no** - enable the error stop function.
- **Make report: yes** – enable the option of creating a checking results log.
- **Make statistics: yes** – you can also enable the option of generating statistical data on the project (if you are interested in this function).

On completion of the procedure in case of any error revealed the system will suggest that you review a text file containing a list of the errors found. (By default this information is moved to the end of the file “checkbkl.lst” in the current folder, but you can specify any other file).

If such errors exist, proceed to the steps described in the following section.

CORRECTING ERRORS IN

ERRORS THAT OCCUR DURING THE OPERATION

The procedure of correcting the errors that occur during the system operation is absolutely similar to the procedure of correcting the data base errors. To correct an error you are required to submit a detailed description of the error and instruction on how to repeat it.

Add summary information on the error to the file *aperrors.lst* located in the *SEA SOLUTION* directory with your comments to the description of the error that results in quitting the system. This file may be of use for debugging.

Quit the *SEA SOLUTION* system and save the current state of the project (file **.bkl**) and the previous version (file **.bak**). You better backup them or copy to another folder. They should not be altered before the error is fully investigated.

Load the project file **.bkl** and check it for correctness (see above). If the system finds an error, first process it as described in the previous section.

Try to repeat the error, sequentially performing the operations during which the error occurred. On finding the operation that results in the error try to perform in under similar conditions in a new project.

As a result you should describe the method of obtaining the error. If you failed to repeat it in a new project or the error is not constant, attach corresponding files to the error: **.bkl** and **.bak**.

Processing of the error is appropriate if it can be repeated according to your description in the laboratory environment (theoretically, with the minimal number of operations and objects in the project).

If the error cannot be repeated, describe all the operations you have performed in the project and which might cause the error, as well as attach your files to the error.

Send you description of the error together with the file “aperrors.lst” (and, if required, your project file) to “Sea Tech” Ltd. for correction.

OPERATION WITH THE "SURFACE" LEVEL DISABLED

In SEA SOLUTION operation with any element requires a corresponding level to be enabled, except for the surface.

With the surface levels disabled ("Surface", "Sections", "Contour") you can add and adjust the surface, specifying the working surface, introduce superficial lines and convert spatial points and lines into superficial points and lines ("Edit/Weld"), as well as perform other commands.