# Shape Maker v2.0

## MSI

The User Manual contains all essential information for the user to make full use of the information system. This manual includes a description of the system functions and capabilities, contingencies and alternate modes of operation

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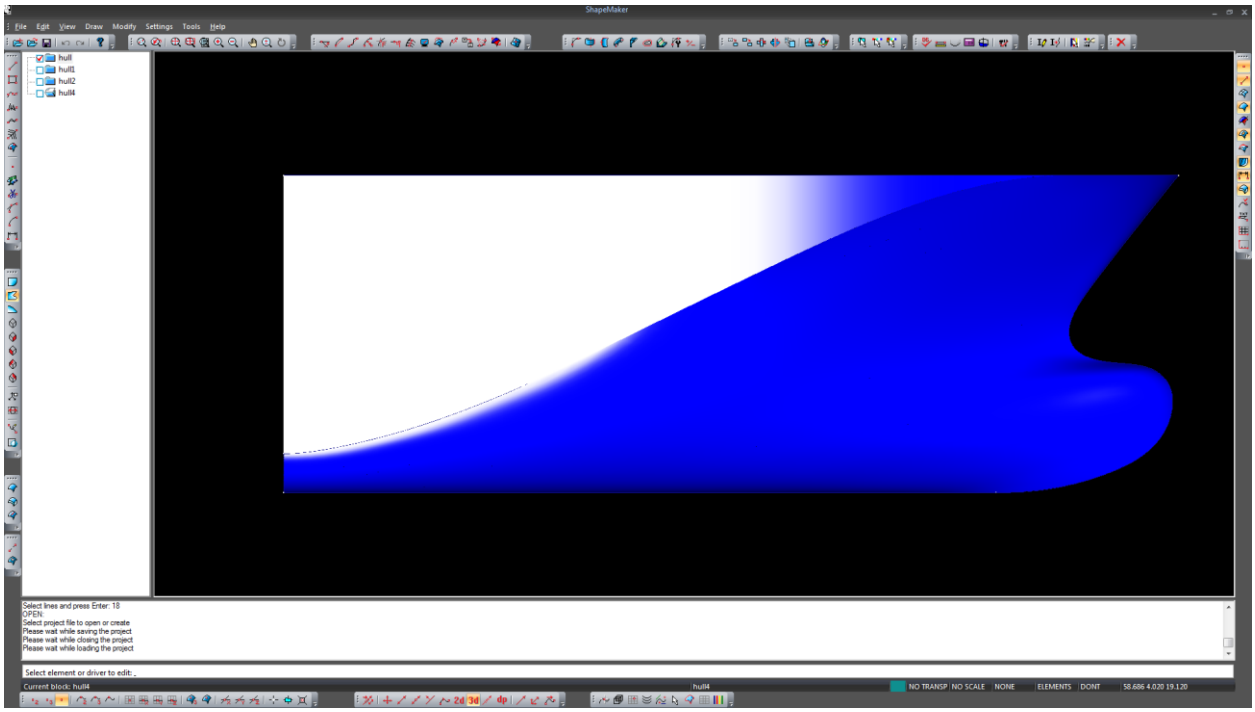
# Table of Contents

# Interface

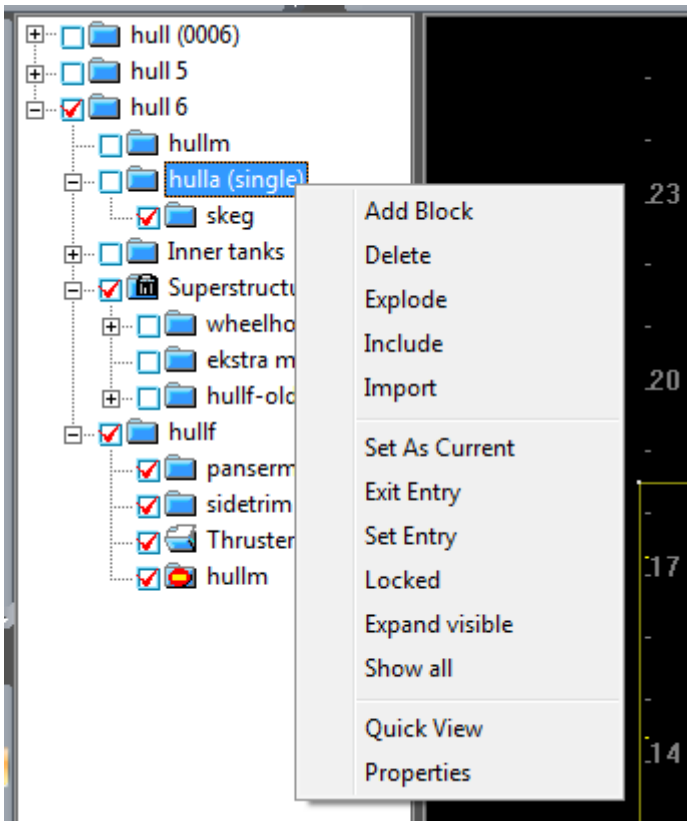
Interface consist of menu, side tree window, message window, command window and status bar and tool bars. All interface elements can be controlled by View menu command.

Recommended toolbars layout:



# Side tree window

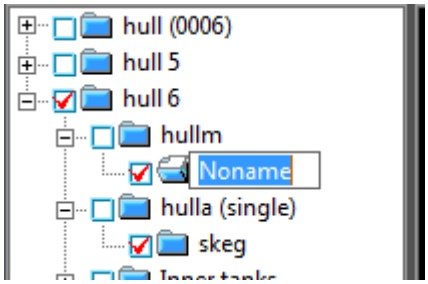
This window used for control of visibility blocks, layer and colors in database.



Access to block's available commands provided by clicking with right mouse button on the block's name.

## Tree ▸ Add Block

This command creates a new empty "Noname" block. Block is become as current. Name can be changed any time. Position of the block can be changed by drag and drop.




## Tree ▸ Delete

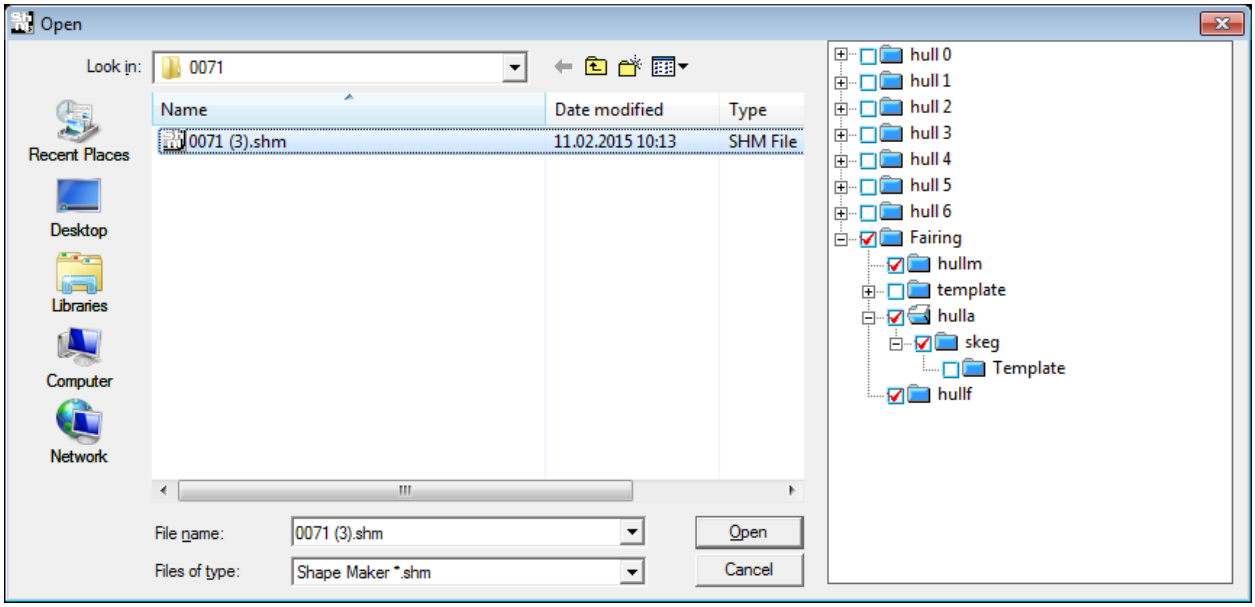
This command delete selected block with all elements from database.

## Tree ▸ Explode

This command delete selected block and move all elements to parent block.

## Tree ▸ Include

Block from external file can be linked to current database by this command. Such block is use only for visualization. To include block is necessary select external database and external block.




After that insert block to required position.

## Tree ▸ Import

Import command is similar to Include with only one difference – all elements from selected block will be copied into current database.

## Tree ▸ Set As Current

Makes selected block current for all new inputted elements. To set block current user can also click with left button on block icon in the tree.

### Tree ▸ Exit Entry
Cancel entry point settings for tree.

### Tree ▸ Set Entry
Set up entry point for the tree. It helps to work with some part of thee as a separate tree.

### Tree ▸ Locked
Lock / Unlock selected block for the editing. To set block Lock / Unlock user can also click with right button on block icon in the tree. Current block can’t be locked.

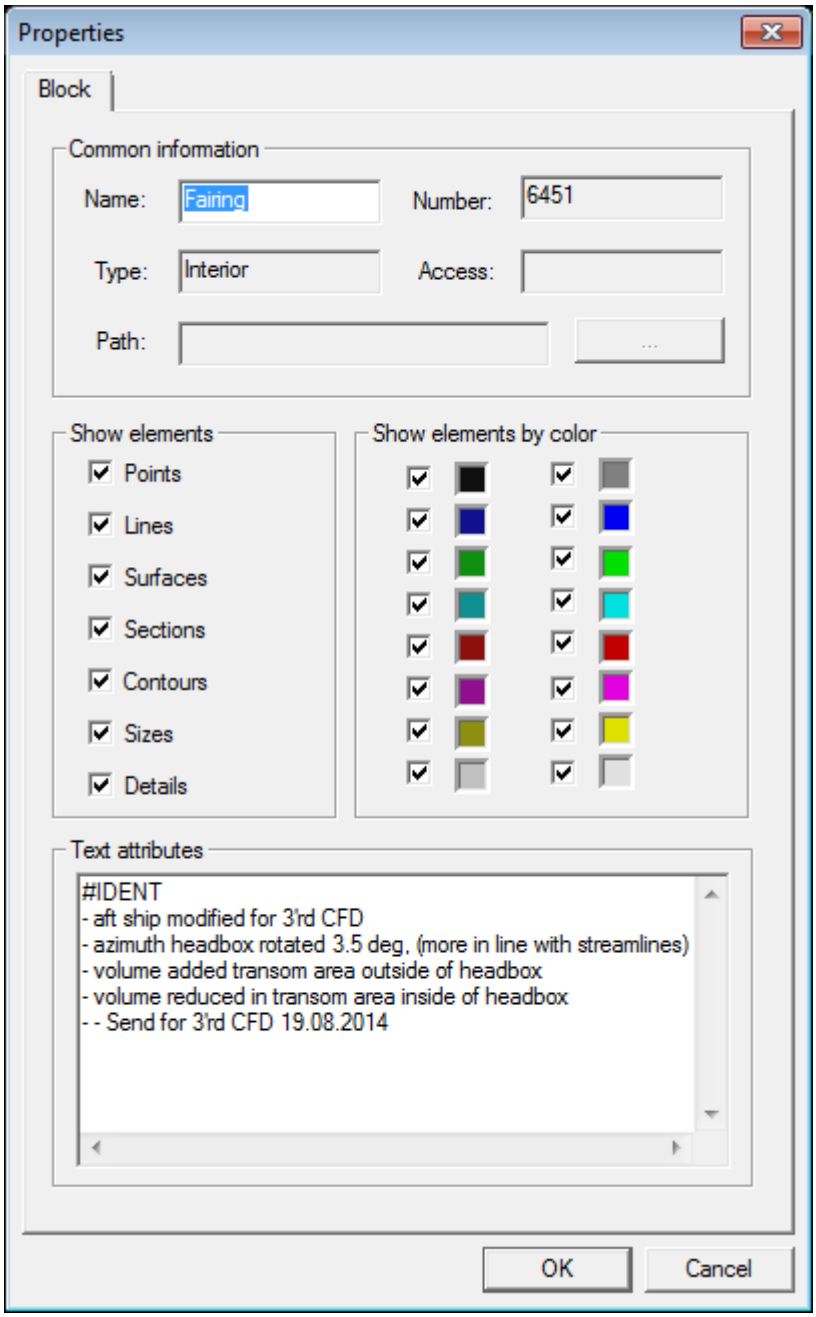
### Tree ▸ Expand Visible
Expand all visible blocks in the tree.

### Tree ▸ Show All
Show all database elements as visible.

### Tree ▸ Quick View
Show selected block in the new window.

## Tree ▸ Properties

This feature allows viewing properties and adjusting visibility of the elements by their properties.




User can determine which elements in the block will be displayed.

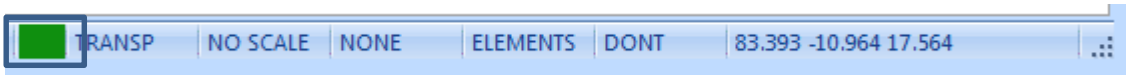
Show elements by color–determine the elements in the block will be displayed.

Text attributes – Additional text information belongs to the block.

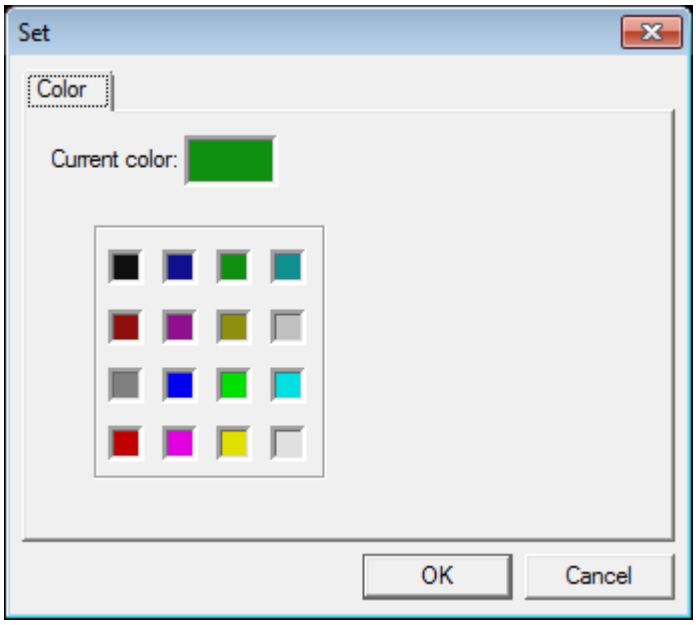
# Status Bar

Status bar fields not only indicate current settings status. They can be also used as hot buttons.

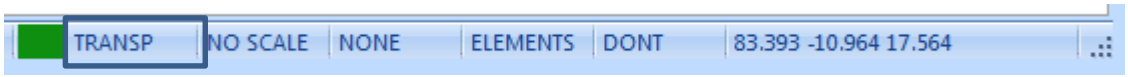
## Status Bar ▸ Color




This field indicated current color. By clicking on the filed user can change current color in following menu:

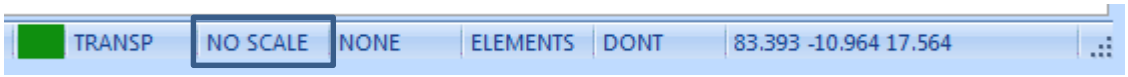



## Status Bar ▸ Transparency



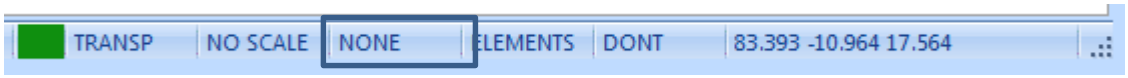

This field indicated transparency status for surfaces.. By clicking on the filed user can change current transparency status.

## Status Bar ▸ Scale

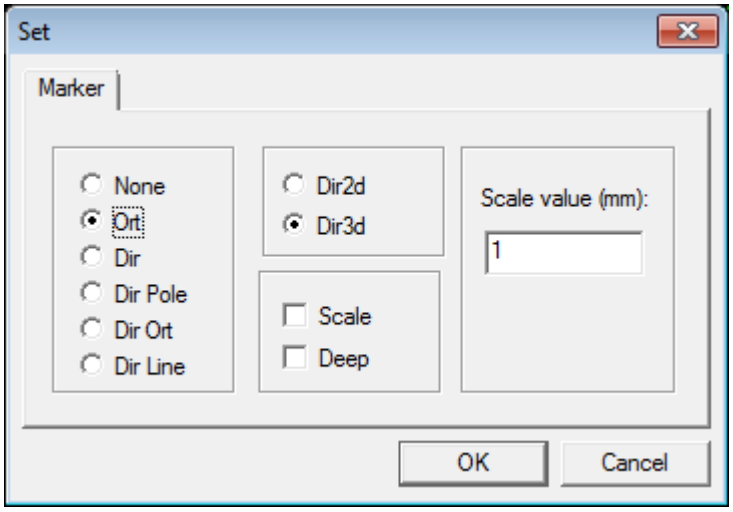



This field indicated scale status for model visualization. By clicking on the filed user can change current scale status.

## Status Bar ▸ Cursor

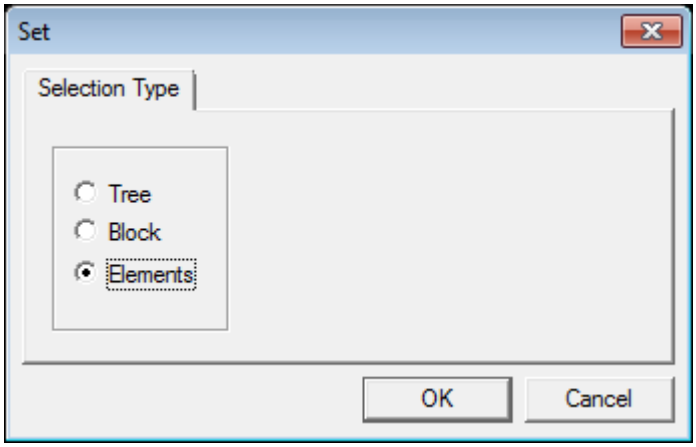



This field indicated cursor movement type. By clicking on the filed user can change it in following menu:

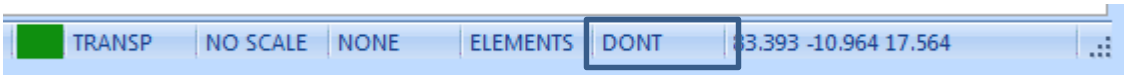



## Status Bar ▸ Selection Type

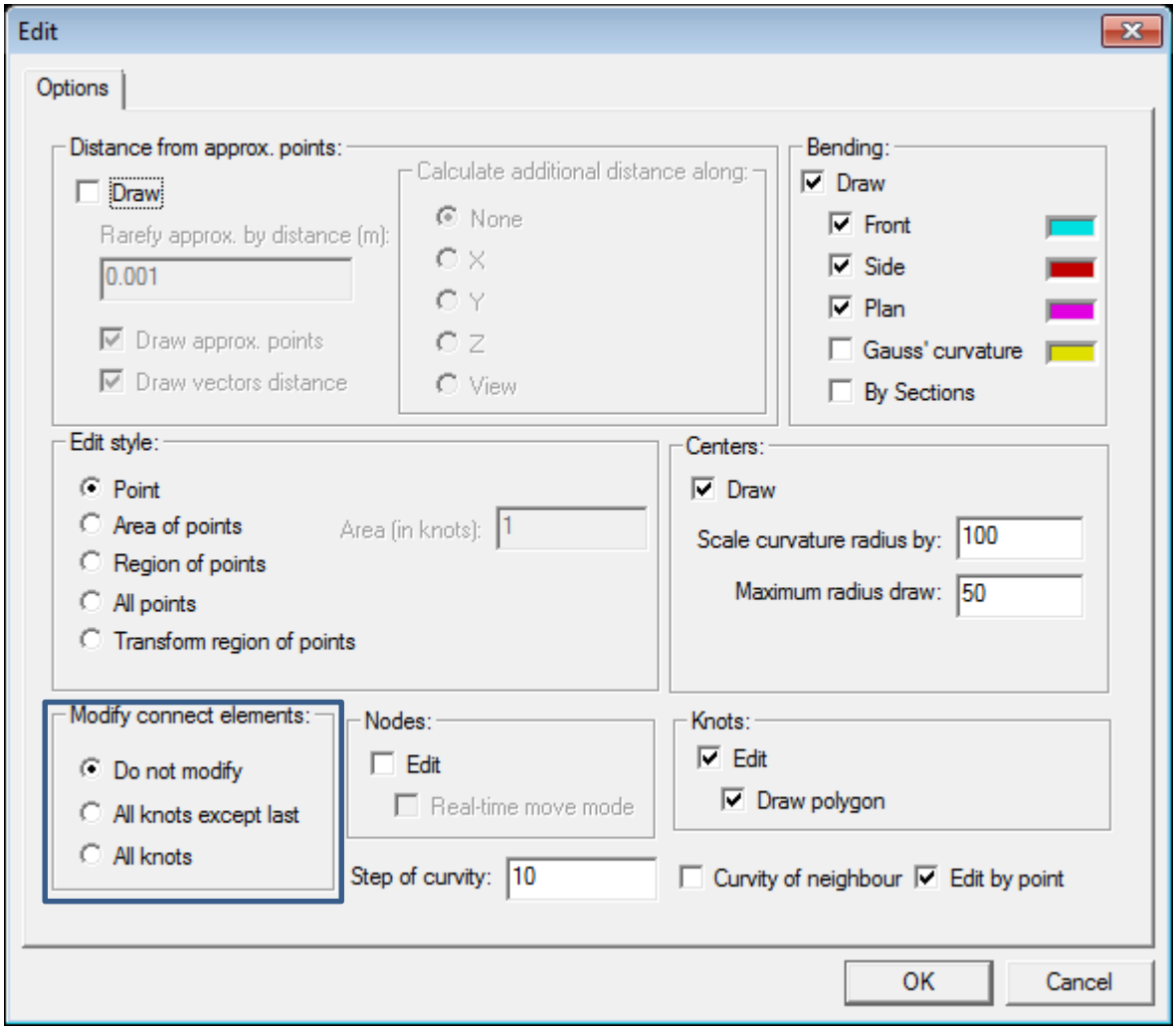

This field indicated cursor selection type. By clicking on the filed user can change it in following menu:



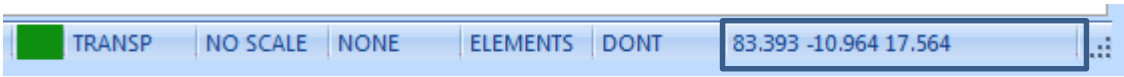
### Status Bar ▸ Selection Type




This field indicated shape modification  type. By clicking on the filed user can change it in following menu:




### Status Bar ▸ Coordinates




This field show the current cursor coordinates.

# Message window

This window used for keep log information of the current working session.

# Command window

This window used for input coordinates during working session.

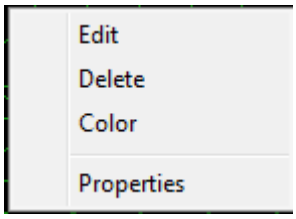
# Toolbars

Toolbars is used as an alternative interface for menu commands. All commands can be run from toolbars and from menu. Toolbars fully support Microsoft Windows interface standards.

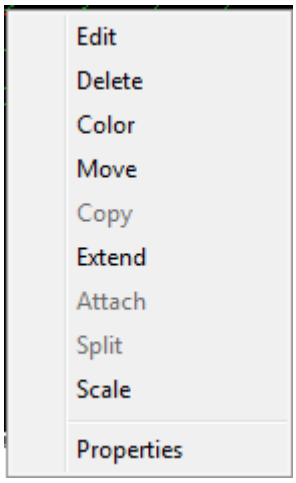
# Context menu

Context menu will appear on some of elements by pointing of the element with right mouse button and Ctrl button. In general context menu consist of most usable commands for the element.

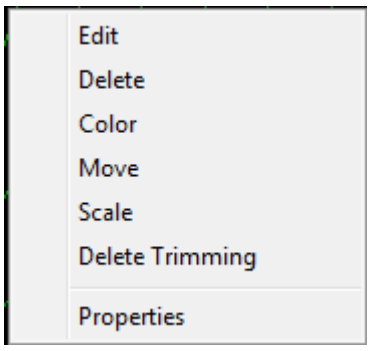
Context menu for point:



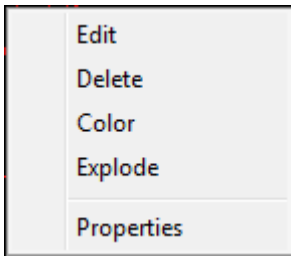

Context menu for line:




Context menu for surface:



Context menu for driver:




# Hot keys

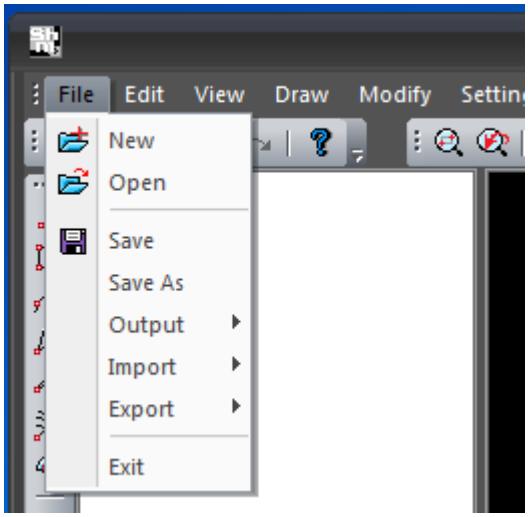
| | |
|---|---|
| Alt X, Alt F4 | File / Exit |
| F2 | File / SaveAs |
| F3, Ctrl O | File / Open |
| Ctrl C | File / Config |
| Ctrl S | Save |
| Ctrl P | Filter / Point |
| Ctr lL | Filter / Line |
| Ctrl G | Filter / Grid |
| Ctrl X | Filter / GridX |
| Ctrl Y | Filter / GridY |
| Ctrl Z | Filter / GridZ |
| 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 |
| Alt F1 | View / Front |
| Alt F2 | View / Side |
| Alt F3 | View / Plan |
| Alt F4 | View / Isometry |
| Alt F5 | Set / Isometry |
| Alt W | Window / New |

| | |
|---|---|
| Alt Q | Window / Prev |
| Alt S | Window / Shift |
| Alt A | Window / All |
| Alt D | Window / Default |
| Alt N | Window / Name |
| Alt BackSpace | Undo |

# Menu

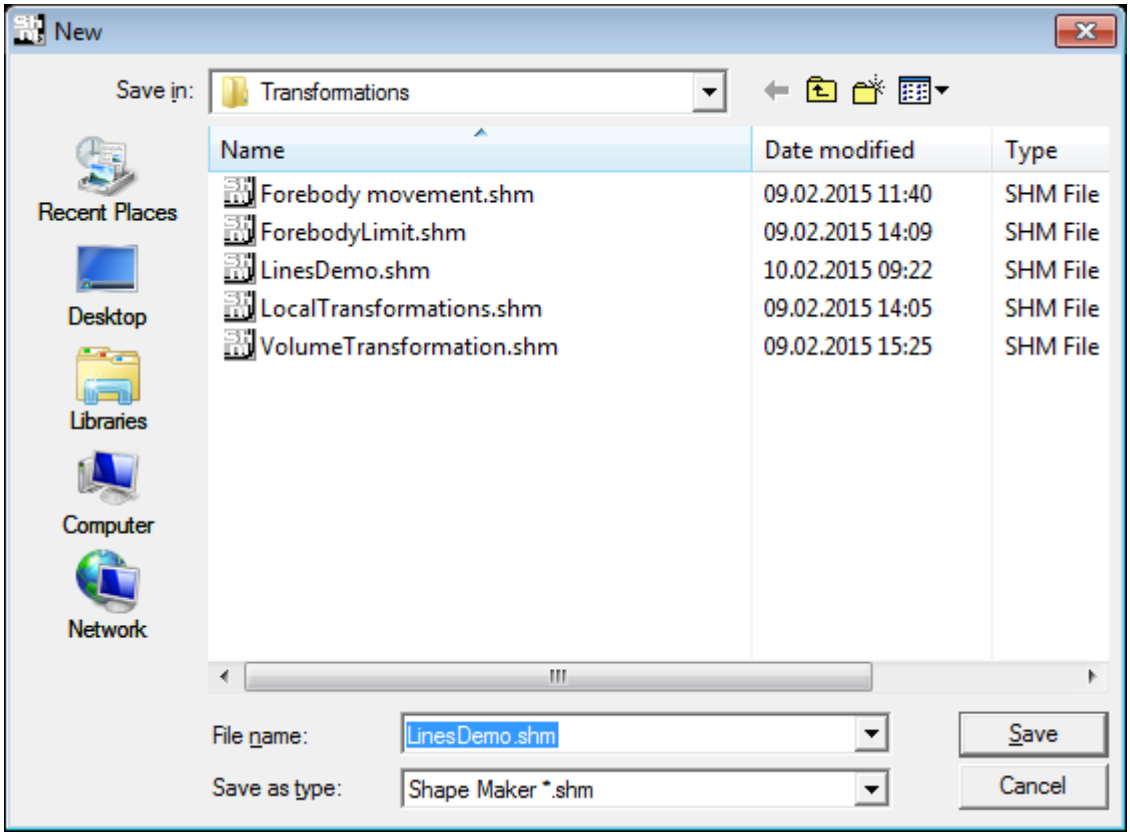
## File

File Menu contains the file management commands, which allow handling the project files, prepare the output documentation files, import or export the files.




### File ▸ New

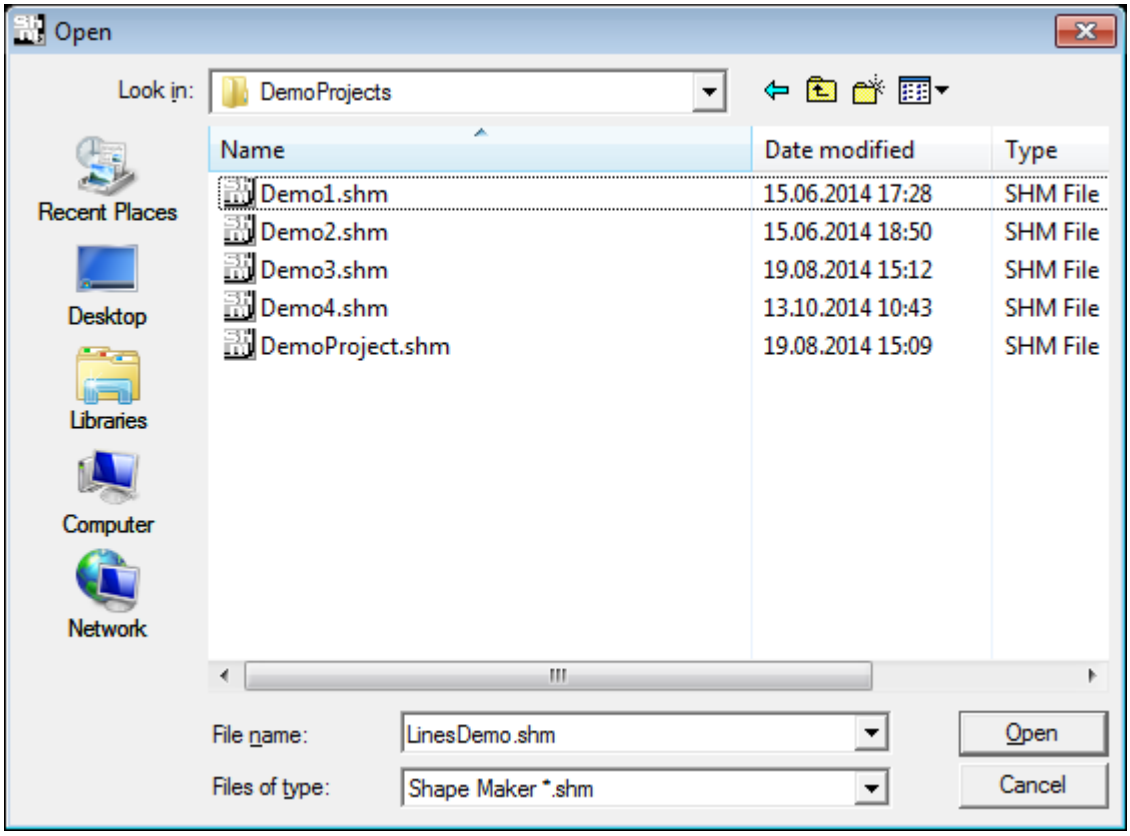
This command allows creating a new project. Follow the dialog to create a new project.




The new project will be opened on the screen with default project settings.

### File ▸ Open

This command is used to open an existing project. In the dialogue box select the project file to be opened.




The project will be opened on the screen to the state it was saved before.

## File ▸ Save

This command allows saving changes in the current project.

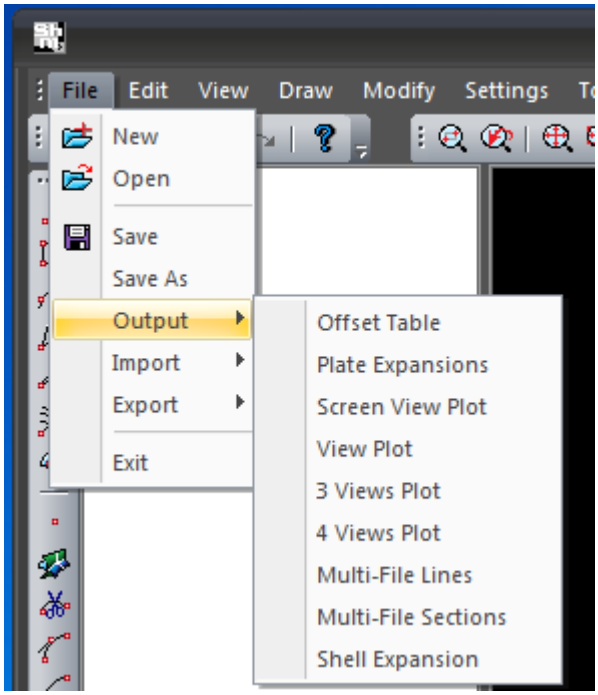
Note: After saving user is not able to use Undo / Redo commands for saved changes.

## File ▸ Save As

This command allows saving the current project with a different name.

## File ▸ Output

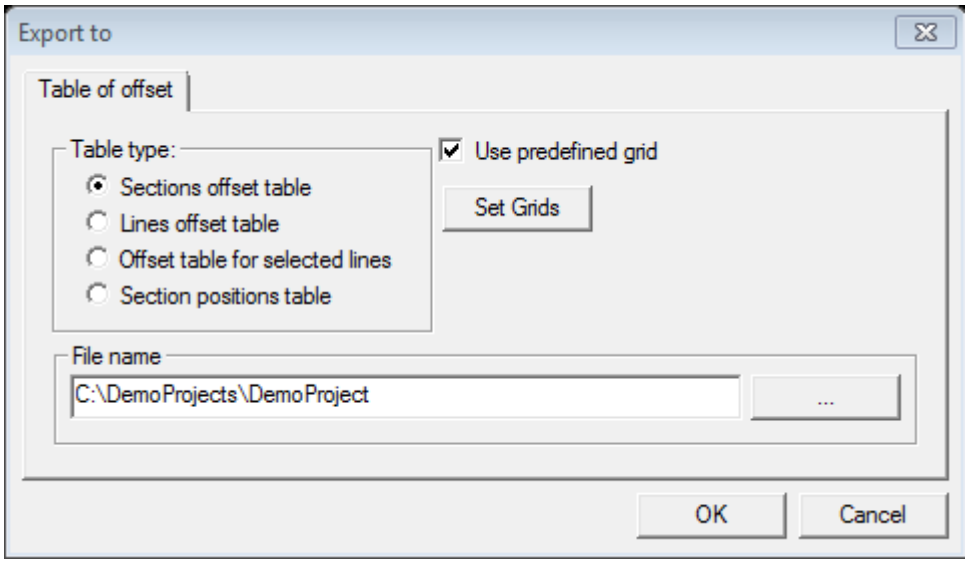
Output menu is used for generation of the project documentation. The output documentation files are transferred in the DXF and TXT file formats.




With this item Shape Maker will generate set of text documents and sketches in according with predefined templates.

**Output ▸ Offset Table**

This command creates an offset table for all turn on objects of the model.




For export to offset table can be chosen 4 different variants of the offset table:

- Section offset table – hull surface sections point's coordinates in according to defined grid.
- Lines offset table – lines point's coordinates in according to defined grid. The offsets of the all lines with #IDENT attributes will be created (buttock in CL, knuckle lines, lines of seems and buts and etc.). The important points of the lines will be transferred additionally (the initial and end points, knuckle points and other). To get more information how to set attributes see attribute command.
- Offset table for selected lines – same as previous, but lines can be selected in the interactive mode. The processed lines are specified by the user. User have to select begin and end points then select way line to define lines chain if it necessary. After that enter text identification of the line.
- Section positions table – coordinates of defined grid sections.

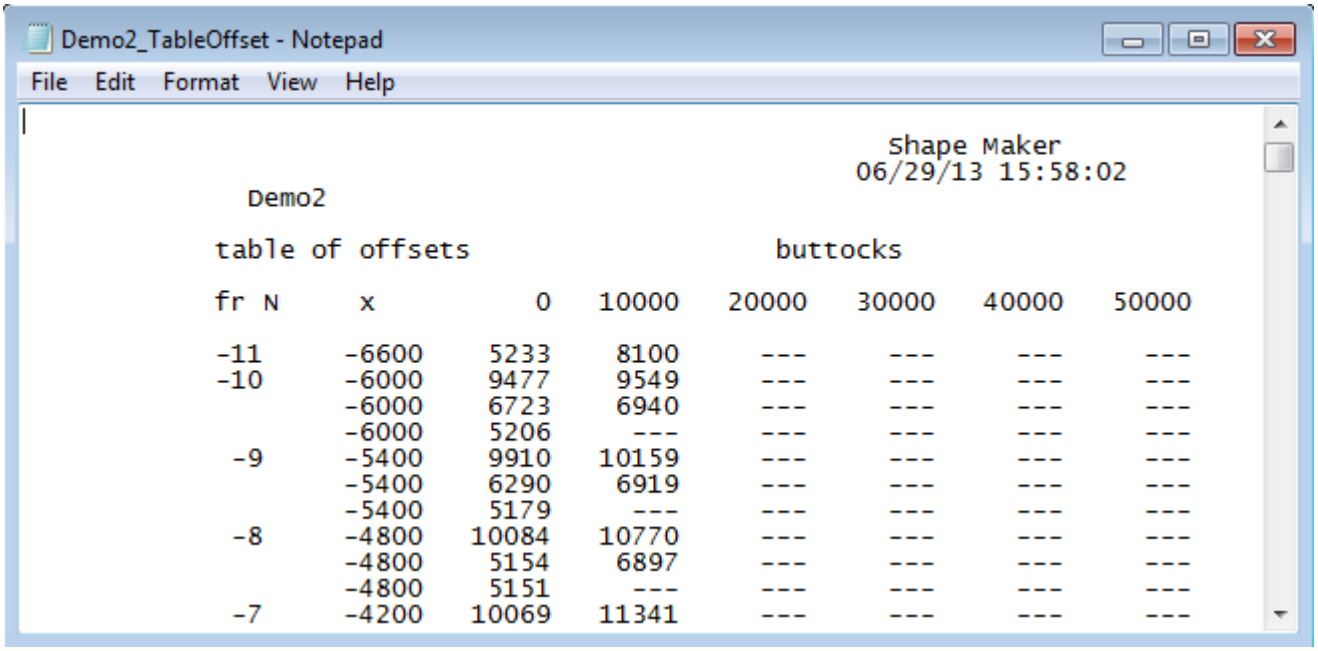
Two variants of the grid can be used:

- Same grid as set for model visualization (Use predefined grid-off).
- Grid defined for offset tables only (Use predefined grid-on).

Grid defined for offset table can be set by clicking Set Grids button.

Shape Maker will generate following documents:

- Section offsets table.



Demo2_TableOffset - Notepad

File Edit Format View Help

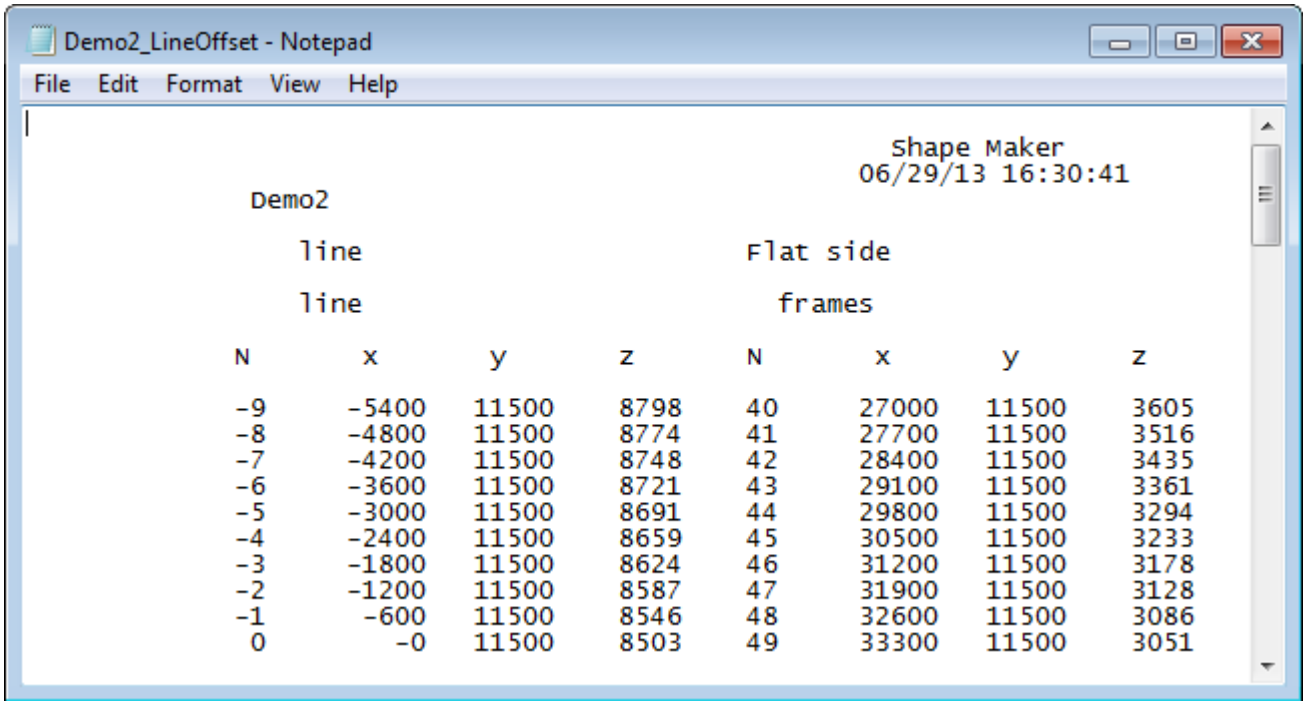
Shape Maker
06/29/13 15:58:02

Demo2

table of offsets buttocks

| fr | N | x | 0 | 10000 | 20000 | 30000 | 40000 | 50000 |
|---|---|---|---|---|---|---|---|---|
| | -11 | -6600 | 5233 | 8100 | --- | --- | --- | --- |
| | -10 | -6000 | 9477 | 9549 | --- | --- | --- | --- |
| | | -6000 | 6723 | 6940 | --- | --- | --- | --- |
| | | -6000 | 5206 | --- | --- | --- | --- | --- |
| | -9 | -5400 | 9910 | 10159 | --- | --- | --- | --- |
| | | -5400 | 6290 | 6919 | --- | --- | --- | --- |
| | | -5400 | 5179 | --- | --- | --- | --- | --- |
| | -8 | -4800 | 10084 | 10770 | --- | --- | --- | --- |
| | | -4800 | 5154 | 6897 | --- | --- | --- | --- |
| | | -4800 | 5151 | --- | --- | --- | --- | --- |
| | -7 | -4200 | 10069 | 11341 | --- | --- | --- | --- |

- Lines offsets table.



Demo2_LineOffset - Notepad

File Edit Format View Help

Shape Maker
06/29/13 16:30:41

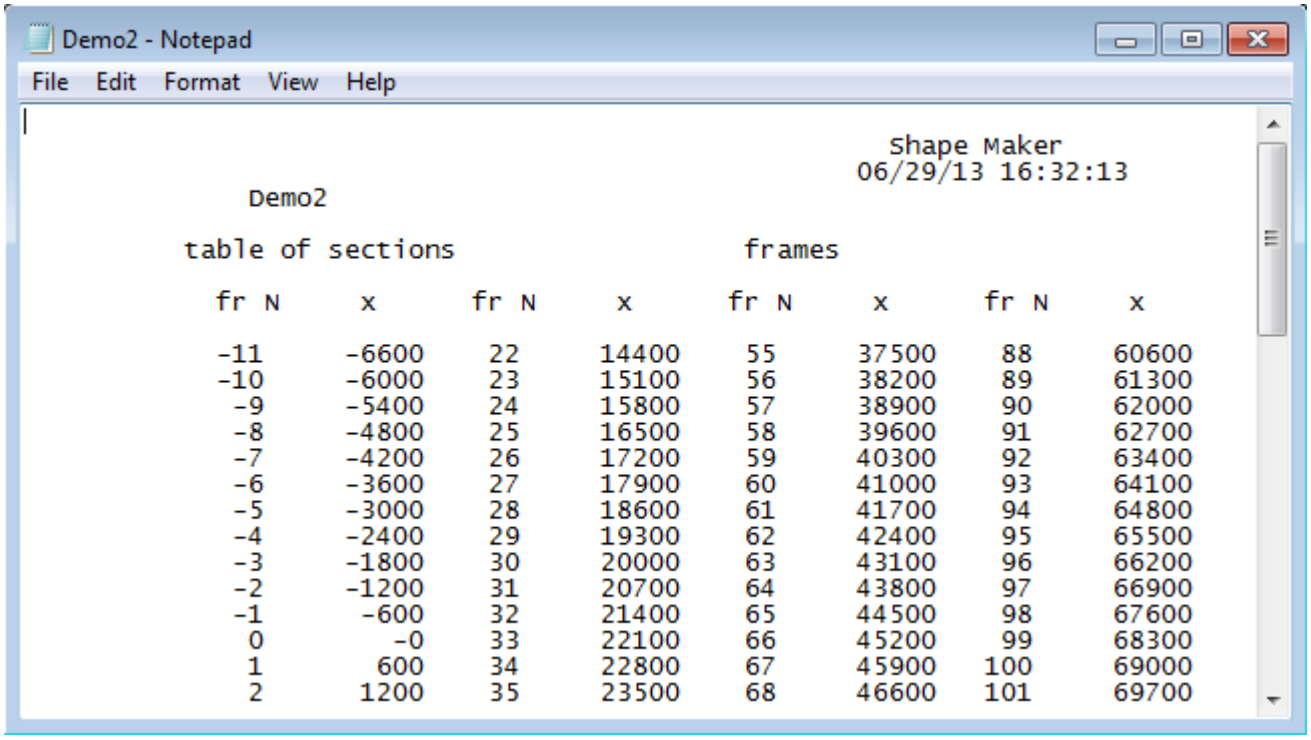
Demo2

line Flat side

line frames

| N | x | y | z | N | x | y | z |
|---|---|---|---|---|---|---|---|
| -9 | -5400 | 11500 | 8798 | 40 | 27000 | 11500 | 3605 |
| -8 | -4800 | 11500 | 8774 | 41 | 27700 | 11500 | 3516 |
| -7 | -4200 | 11500 | 8748 | 42 | 28400 | 11500 | 3435 |
| -6 | -3600 | 11500 | 8721 | 43 | 29100 | 11500 | 3361 |
| -5 | -3000 | 11500 | 8691 | 44 | 29800 | 11500 | 3294 |
| -4 | -2400 | 11500 | 8659 | 45 | 30500 | 11500 | 3233 |
| -3 | -1800 | 11500 | 8624 | 46 | 31200 | 11500 | 3178 |
| -2 | -1200 | 11500 | 8587 | 47 | 31900 | 11500 | 3128 |
| -1 | -600 | 11500 | 8546 | 48 | 32600 | 11500 | 3086 |
| 0 | -0 | 11500 | 8503 | 49 | 33300 | 11500 | 3051 |

- Sections table.



Demo2 - Notepad

File Edit Format View Help

Shape Maker
06/29/13 16:32:13

Demo2

table of sections frames

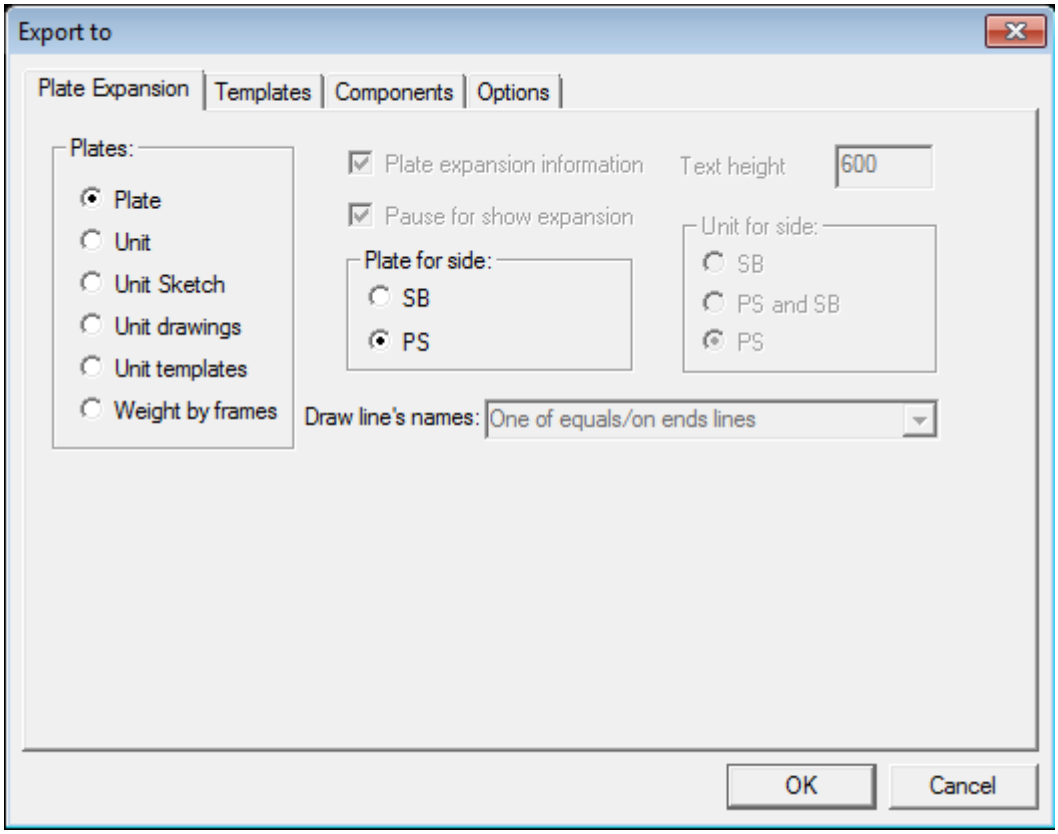
| fr N | x | fr N | x | fr N | x | fr N | x |
|---|---|---|---|---|---|---|---|
| -11 | -6600 | 22 | 14400 | 55 | 37500 | 88 | 60600 |
| -10 | -6000 | 23 | 15100 | 56 | 38200 | 89 | 61300 |
| -9 | -5400 | 24 | 15800 | 57 | 38900 | 90 | 62000 |
| -8 | -4800 | 25 | 16500 | 58 | 39600 | 91 | 62700 |
| -7 | -4200 | 26 | 17200 | 59 | 40300 | 92 | 63400 |
| -6 | -3600 | 27 | 17900 | 60 | 41000 | 93 | 64100 |
| -5 | -3000 | 28 | 18600 | 61 | 41700 | 94 | 64800 |
| -4 | -2400 | 29 | 19300 | 62 | 42400 | 95 | 65500 |
| -3 | -1800 | 30 | 20000 | 63 | 43100 | 96 | 66200 |
| -2 | -1200 | 31 | 20700 | 64 | 43800 | 97 | 66900 |
| -1 | -600 | 32 | 21400 | 65 | 44500 | 98 | 67600 |
| 0 | -0 | 33 | 22100 | 66 | 45200 | 99 | 68300 |
| 1 | 600 | 34 | 22800 | 67 | 45900 | 100 | 69000 |
| 2 | 1200 | 35 | 23500 | 68 | 46600 | 101 | 69700 |

**Output ▸ Plate Expansions**

This command used for output shell plate development information, sketches, templates and geometrical characteristics.

Shape Maker has following output variants:

- Plate expansion.




Plates:

Plate – sketches for one plate.

Unit – unit weight characteristics and COG in the TXT and DXF formats, unit sketch and sketches for all shell plates in the DXF file format.

Unit sketch - unit sketch and unit weight characteristics and COG in the TXT and DXF file formats.

Unit drawings – unit sketch in the DXF file format.

Unit templates – shell plates templates for the unit in the DXF file format.

Weight by frames – weight of the unit distributed by frames. Used for weight estimation.

Plate for side:

For the following menu items Plate, Unit, Unit sketch, Unit drawings it is necessary to specify the type of the unit: SB – if unit only on the starboard, PS - the unit is only for the port side. PS and SB – for units symmetrical relatively to CL.

Draw line's names:

Selection of the visualization type for lines with #IDENT attributes.

One of equals/ on ends lines – one identification in the ends of the lines.

One of equals/ on middle lines - one identification in the middle of the line.

All equals/ on ends lines – all identifications in the ends of the lines.

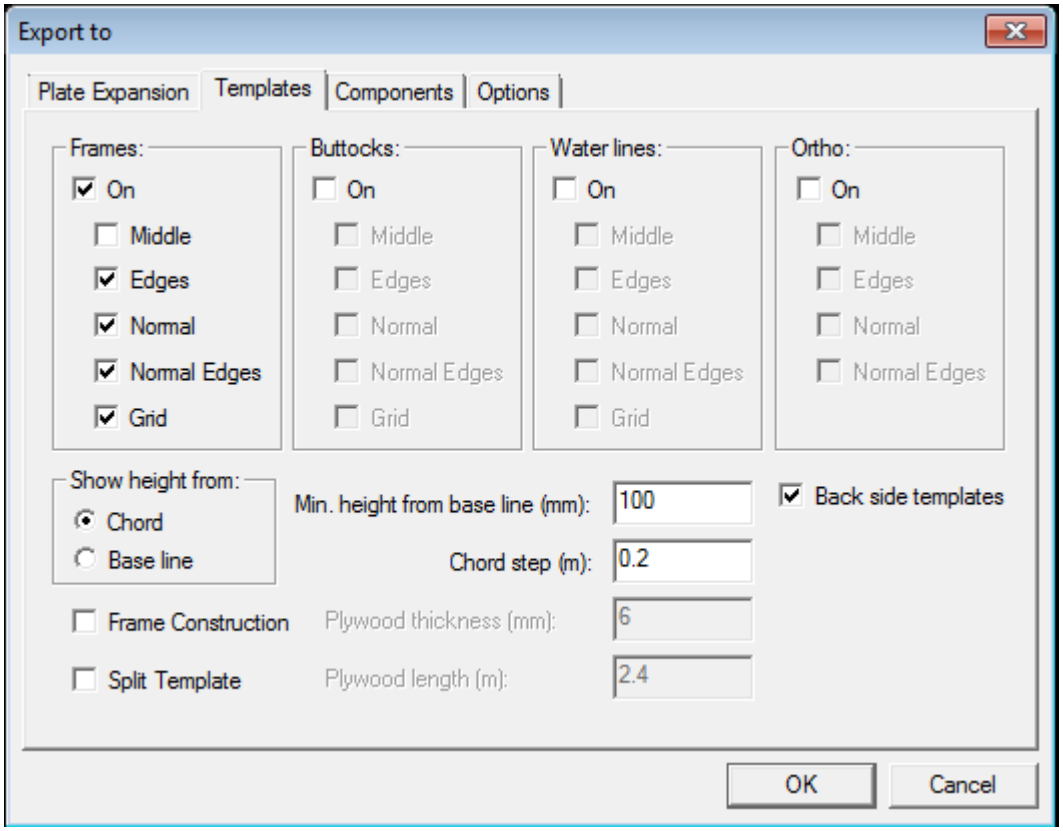
All equals/ on middle lines - all identifications in the middle of the line.

Plate expansion information - show the unit information.

Pause for show expansion - a pause to show each developed shell plate.

Text height (mm) - height of the text in dxf sketches.

- Templates.




Frames – frame templates type definition.

Buttocks – buttock templates type definition.

Water lines – waterline templates type definition.

Cross normal – templates which are in perpendicular to the normal template.

Templates type:

Middle - template passing through the middle of the plate, calculated by the program automatically.

Edges - templates passing at edges of the plate, calculated by the program automatically.

Normal – middle template of normal to surface.

Normal edges - edge template of normal to surface.

Grid - standard grid templates defined by the project grid.

Show height from: type of the basis on which the templates will be created.

Chord - from chord.

Base Line - from the base line.

Min. height from base line (mm) – minimal height from the base line.

Chord step (m):  - chord step to indicate deviations.

Frame Construction - framing construction templates generation.

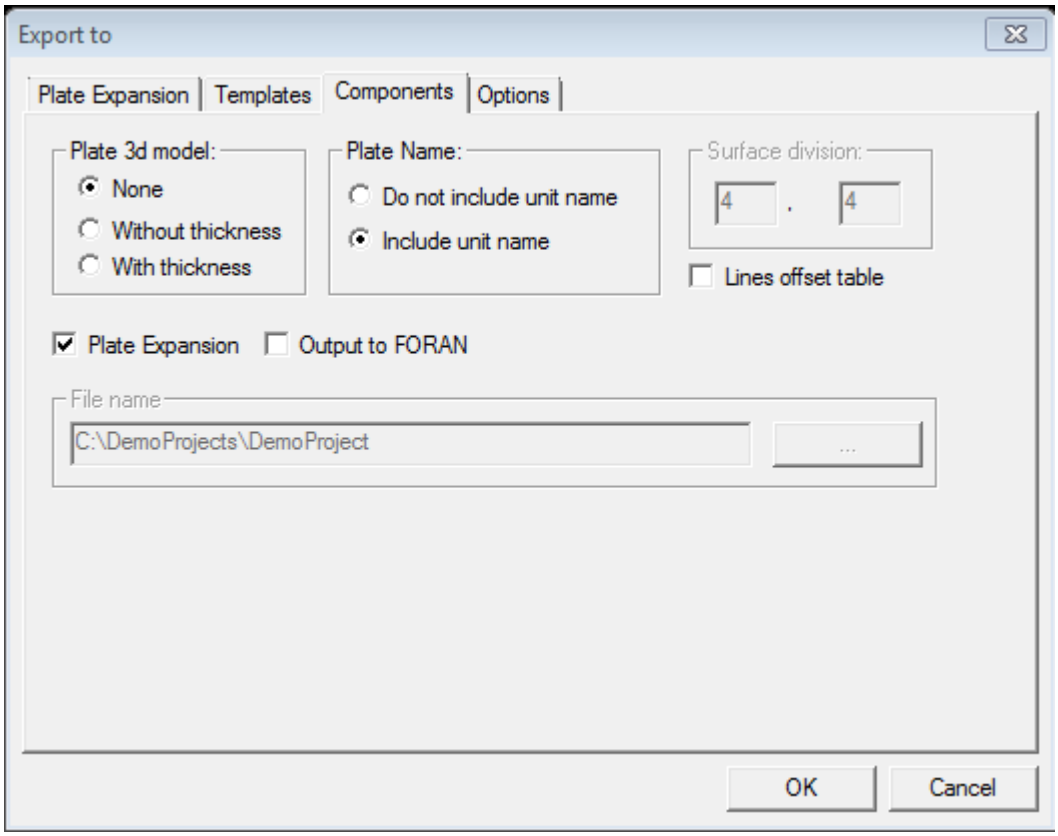
Split Template - templates splitting by maximal length of template's material.

Plywood thickness (mm) – thickness of template material.

Plywood  length (m) - overall length of template material.

Back side templates – templates from the opposite side of the plating.

- Components.




Dialogue box contains options for creating various types of the output files including those in other formats.

Plate 3d model:

None – no output.

Without thickness – without thickness of the shell plates.

With thickness – with thickness of the shell plates.

Plate name:

Do not include unit name – the unit name will not be included in the shell plate file name.

Include unit name – the unit name will be included in the shell plate file name.

Surface division – subdivision surface for faces visualization.

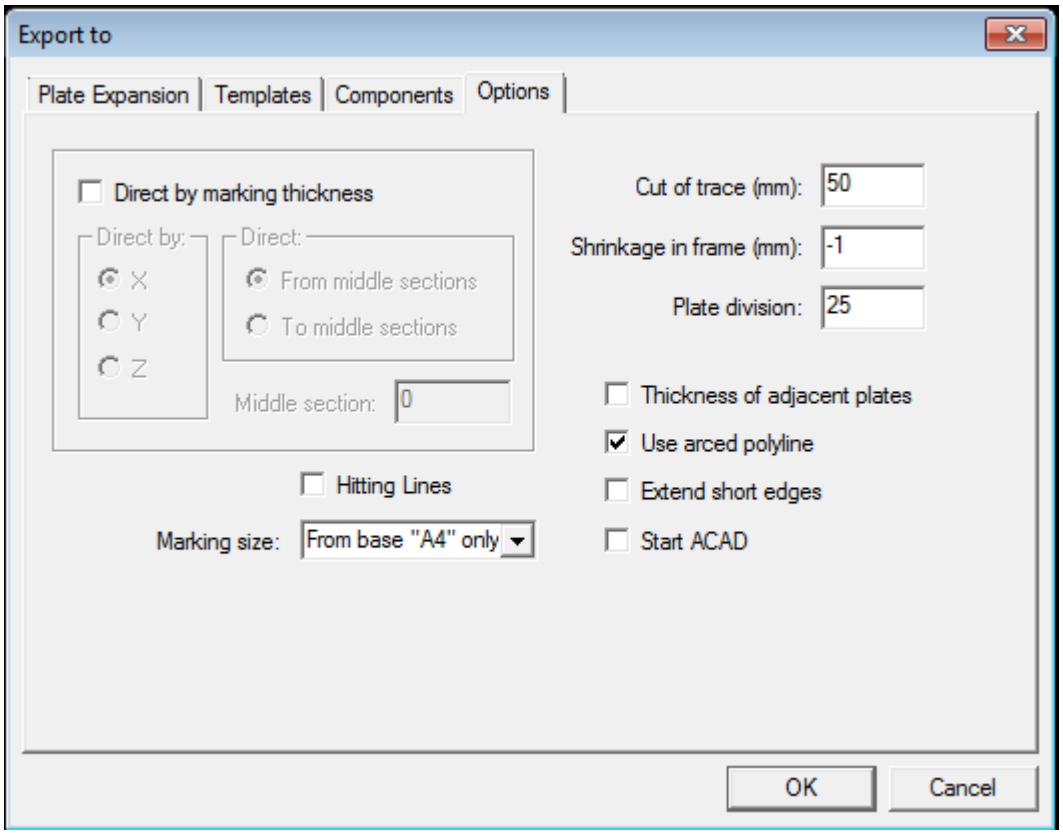
Lines offset table - generation of lines offset table.

Plate Expansion – plate contours sketches generation.

Output to Foran – plate contours file generation in Foran file type.

File name – the name of the output file.

- Options.




Setting up shell plates development parameters.

Direct by marking thickness – indication of the marking line thickness direction in the shell plates sketches.

Direct by:

X – direction of thickness to the bow or stern.

Y – direction of thickness to the side or middle.

Z – upward or downward direction of thickness.

Direct: direction of the framing thickness.

From middle sections – from the middle section

To middle sections – to the middle section.

Middle section – value of the middle section in meters from which direction of the framing thickness will be determined.

Hitting Lines - hitting lines generation for heat bending method.

Marking size: – type of the marking line end size.

From base A4 only – dimension from the base only in A4 format

By chain A4 only – dimension by chain only in A4 format

From base R also – dimension in scale 1:1 from base and in development files.

By chain R also – dimension in scale 1:1 from base and in development files.

Cut of trace (mm) - minimal distance between marking line and plate edge in millimeters.

Shrinkage in frame (mm) –shrinkage value per one frame-spacing in millimeters

Plate division – surface patch division for shell plate development.

Thickness of adjacent plates - thickness of adjacent plates considered.

Use arced polyline - arcs used in the polylines.

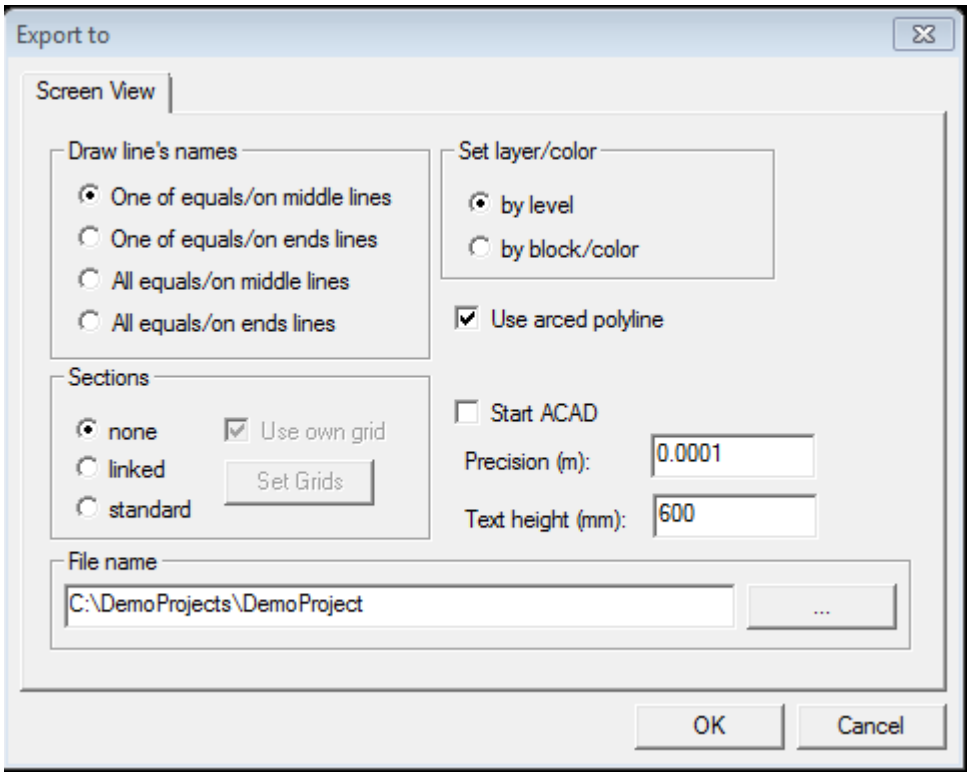
Extend short edges - extend the development edges

Start ACAD - loads the ACAD program after the plate is developed.

**Output ▸ Screen View Plot**

This command creates a copy of the image on the screen in the DXF format.

Visible on the main view part of the project, will be exported to the DXF file.




Draw line's names: - draw line's names lines with #IDENT attributes.

One of equal/on middle lines – draw name for a line in the center.

One of equal/on ends lines – draw name for a line in the ends.

All equals/on middle lines – draw name for a group of lines in the center.

All equals/on ends lines – draw name for a group of lines in the ends.

Sections: - draw sections lines.

none – no sections lines.

linked – draw section lines (one polyline per one section).

standard – draw section lines for each surface separately (one polyline per one surface patch).

Use own grid – enables or disables usage of the own grid.

Set Grids – changes parameters of the own grid.

Set layer/color: colors used when lines transferring.

by level - one color for all elements.

by block/color **-** the color of the project.

Use arced polyline **-** polylines with arcs for lines draw.

Only one arc and one line - Lines and arcs will follow one by one in polyline sequence.

Start ACAD - loads the ACAD program after the plate is developed.

Precision (m) - precision for lines approximation in meters.

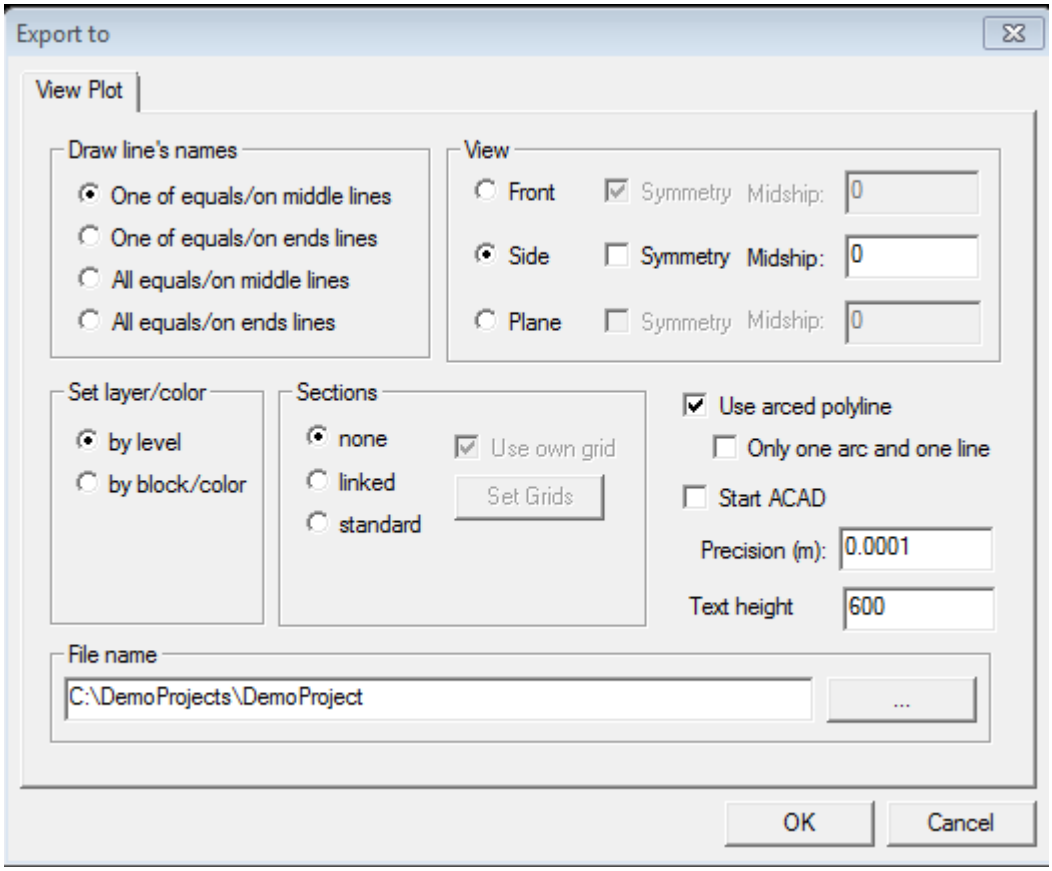
Text height (mm) - height of the lines identification text in millimeters.

File name – the name of the output file.

**Output ▸ View Plot**

This command creates one of the lines drawing views in the DXF file.

Select the lines drawing view options in the dialogue box.




Draw line's names: - draw line's names lines with #IDENT attributes.

One of equal/on middle lines – draw name for a line in the center.

One of equal/on ends lines – draw name for a line in the ends.

All equals/on middle lines – draw name for a group of lines in the center.

All equals/on ends lines – draw name for a group of lines in the ends.

Sections: - draw sections lines.

none – no sections lines.

linked – draw section lines (one polyline per one section).

standard – draw section lines for each surface separately (one polyline per one surface patch).

Use own grid – enables or disables usage of the own grid.

Set Grids – changes parameters of the own grid.

Set layer/color:  colors used when lines transferring.

by level -  one color for all elements.

by block/color **-**  the color of the project.

Use arced polyline **-** polylines with arcs for lines draw.

Only one arc and one line - Lines and arcs will follow one by one in polyline sequence.

Start ACAD -  loads the ACAD program after the plate is developed.

Precision (m) - precision for lines approximation in meters.

Text height (mm) - height of the lines identification text in millimeters.

File name – the name of the output file.

View:  selection of view.

Front – hull view.

Side – side view.

Plane – plan view.

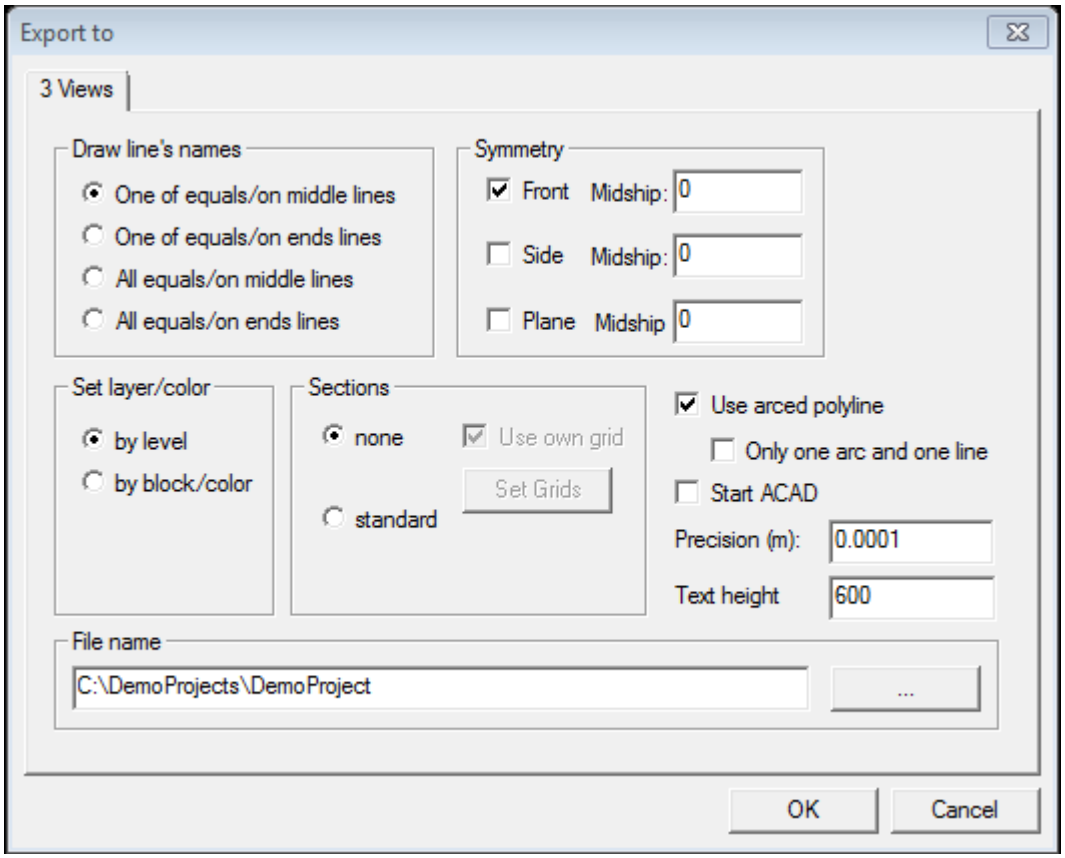
Symmetry – mirroring lines relative of the middle section.

Midship – position of the middle section in meters.

**Output ▸ 3Views Plot**

This command generate a standard lines drawing (three views) in the DXF file.

Visible on the main view part of the project, will be exported to the DXF file.

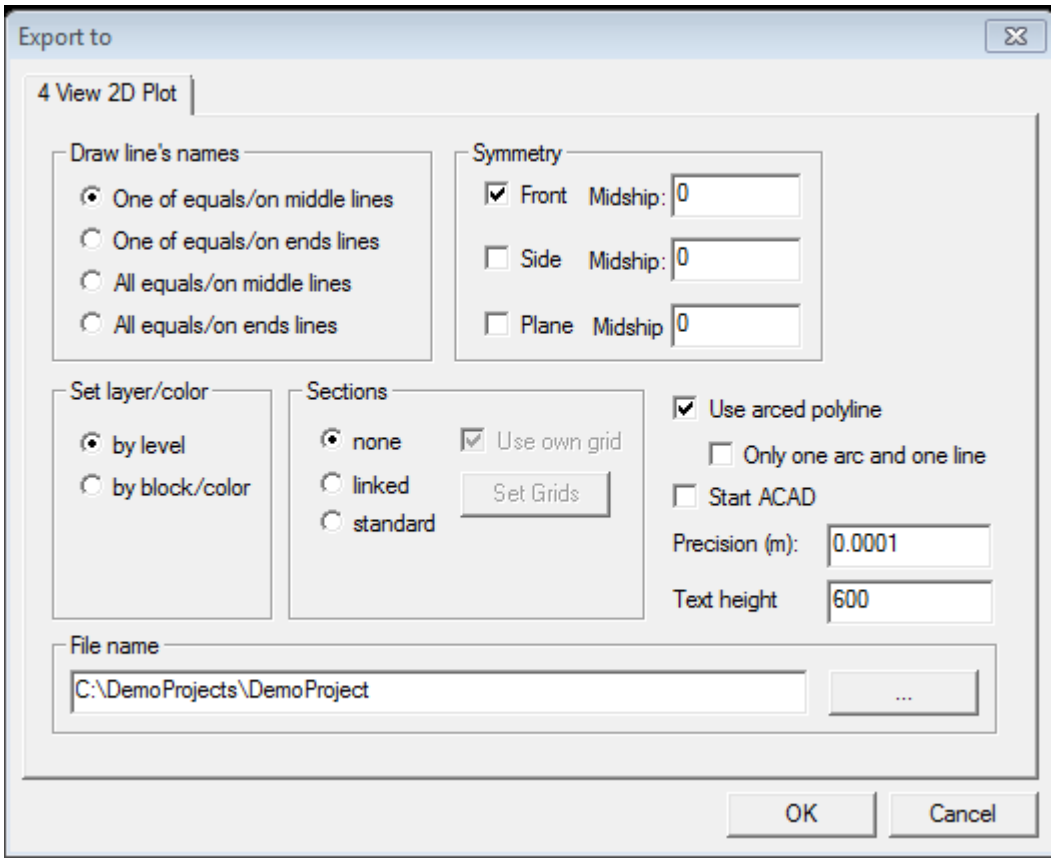



These options are similar to the View Plot command

**Output ▸ 4Views Plot**

This command generate a standard drawing (four views) in the DXF file.

Visible on the main view part of the project, will be exported to the DXF file.

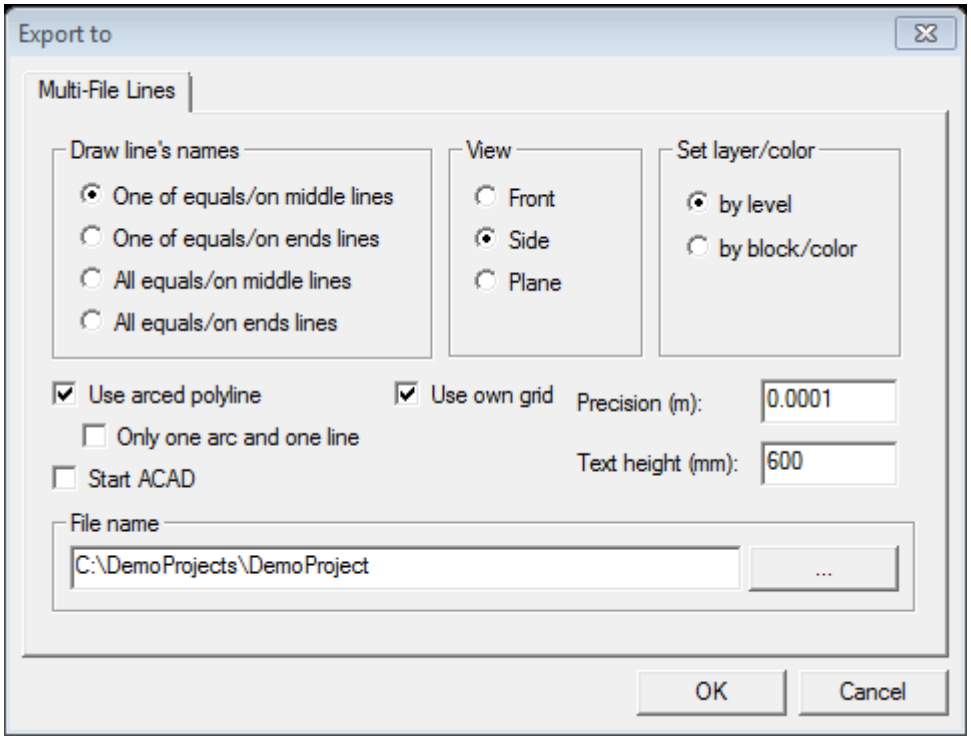



These options are similar to the 3View Plot command. The difference is that two front views present on the drawings - one with stations and another one with building frames.

**Output ▸ Multi-File Lines**

This command generates a line to the DXF file in one of the projections. Each line is saved in the own DXF file.

The file name is name given to the line by line's names lines with #IDENT attributes. To get more information how to set attributes see attribute command.




Draw line's names: - draw line's names lines with #IDENT attributes.

One of equal/on middle lines – draw name for a line in the center.

One of equal/on ends lines – draw name for a line in the ends.

All equals/on middle lines – draw name for a group of lines in the center.

All equals/on ends lines – draw name for a group of lines in the ends.

View**:**

Front – hull view.

Side – side view.

Plane – plan view.

Set layer/color: colors used when lines transferring.

by level - one color for all elements.

by block/color **-** the color of the project.

Use arced polyline **-** polylines with arcs for lines draw.

Only one arc and one line - Lines and arcs will follow one by one in polyline sequence.

Start ACAD - loads the ACAD program after the plate is developed.

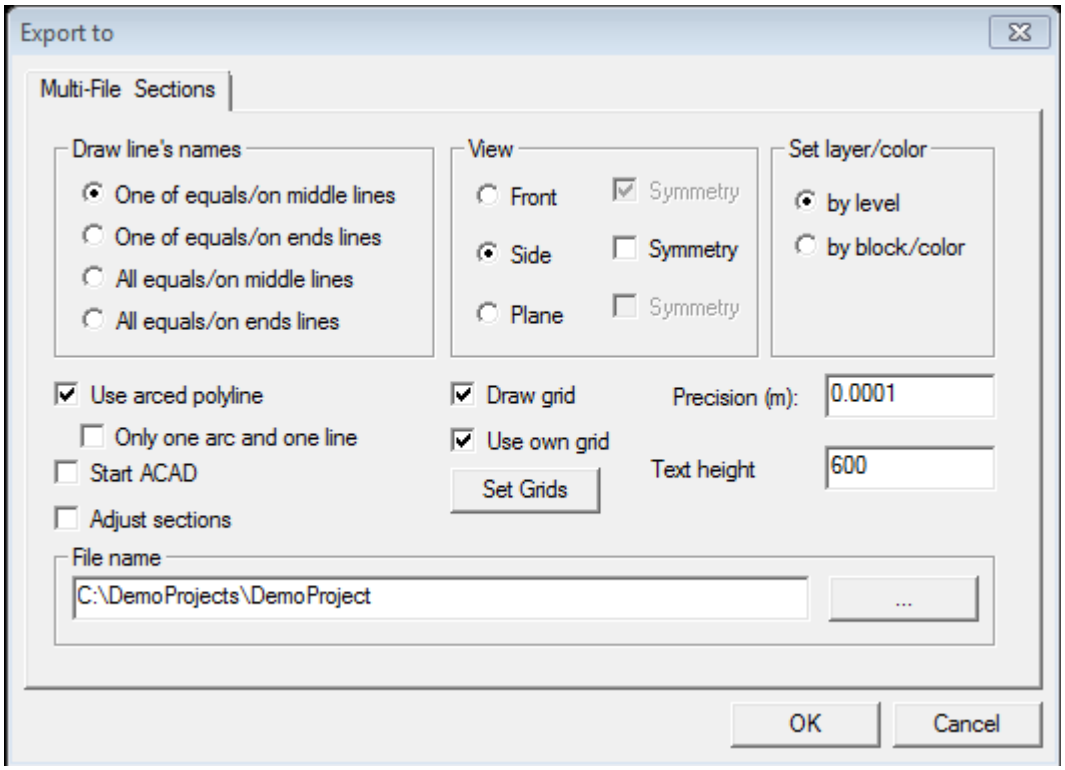
Precision (m) - precision for lines approximation in meters.

Text height (mm) - height of the lines identification text in millimeters.

File name – the name of the output files directory.

**Output ▸ Multi-File Sections**

This command generates a sections to the DXF file in one of the projections. Each section is saved in the own DXF file.




Draw line's names: - draw line's names lines with #IDENT attributes.

One of equal/on middle lines – draw name for a line in the center.

One of equal/on ends lines – draw name for a line in the ends.

All equals/on middle lines – draw name for a group of lines in the center.

All equals/on ends lines – draw name for a group of lines in the ends.

View**:**

Front – hull view.

Side – side view.

Plane – plan view.

Symmetry – mirroring sections.

Set layer/color: colors used when lines transferring.

by level - one color for all elements.

by block/color **-** the color of the project.

Use arced polyline **-** polylines with arcs for lines draw.

Only one arc and one line - Lines and arcs will follow one by one in polyline sequence.

Adjust sections – move zero position of section to ACAD zero position.

Start ACAD - loads the ACAD program after the plate is developed.

Precision (m) - precision for lines approximation in meters.

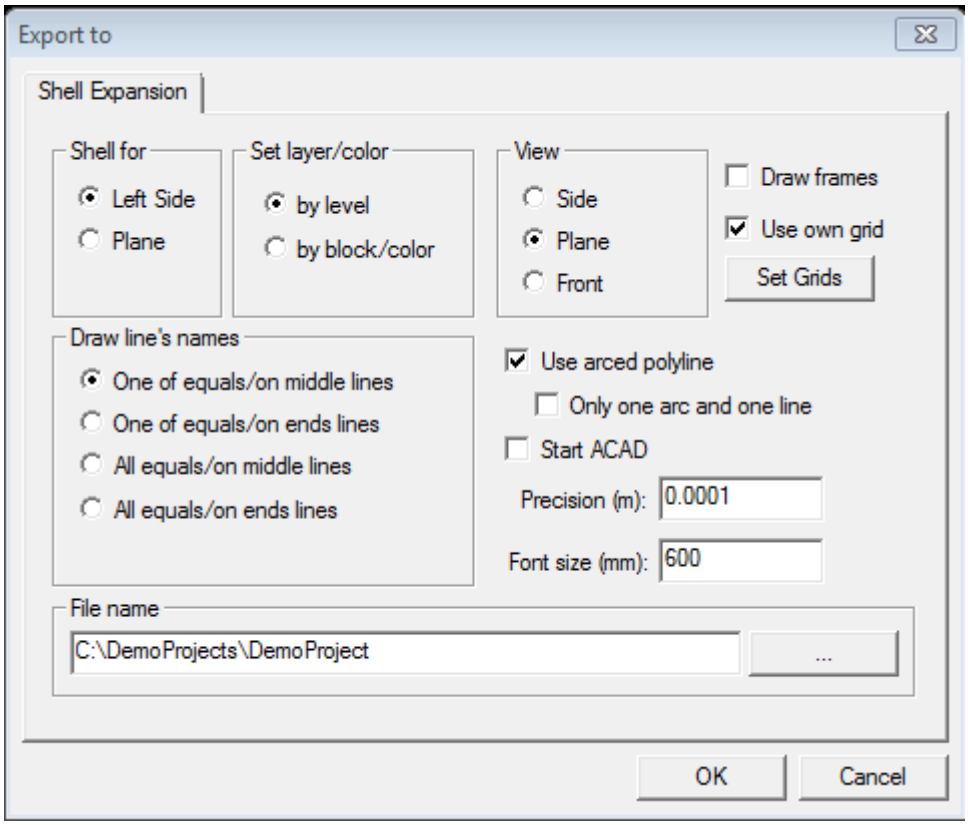
Text height (mm) - height of the lines identification text in millimeters.

File name – the name of the output files directory.

**Output ▸ Shell Expansion**

This command generates a shell expansion drawing.

To develop the shell expansion you need to create the stc file first. The name of the stc file must be same as project file name and located in the same directory.




Shell for:

Left Side – only for the port side.

Plane – for port side and starboard.

Draw frames **-** draw the frame lines.

Set layer/color: colors used when lines transferring.

by level - one color for all elements.

by block/color **-** the color of the project.

View**:**

Front – body plan view.

Side – side view.

Plane – plan view.

Draw line's names: - draw line's names lines with #IDENT attributes.

One of equal/on middle lines – draw name for a line in the center.

One of equal/on ends lines – draw name for a line in the ends.

All equals/on middle lines – draw name for a group of lines in the center.

All equals/on ends lines – draw name for a group of lines in the ends.

Use arced polyline **-** polylines with arcs for lines draw.

Only one arc and one line - Lines and arcs will follow one by one in polyline sequence.

Start ACAD - loads the ACAD program after the plate is developed.

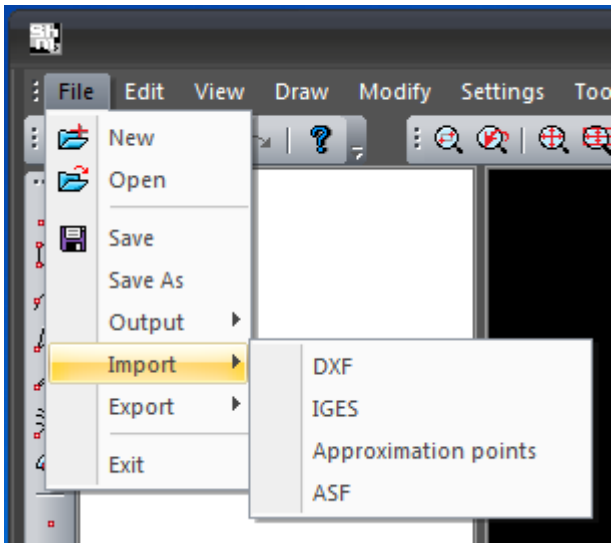
Precision (m) - precision for lines approximation in meters.

Text height (mm) - height of the lines identification text in millimeters.

File name – the name of the output file.

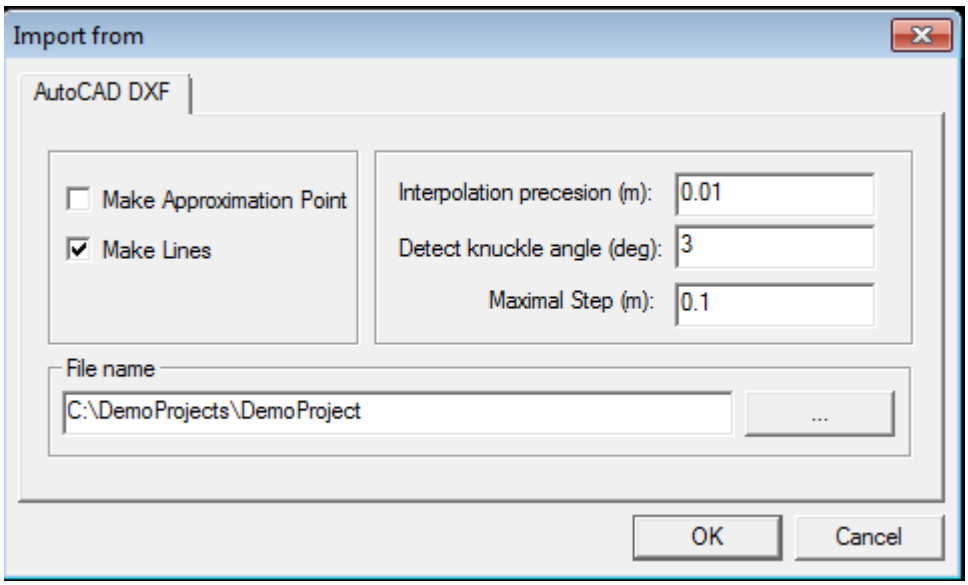
## File ▸ Import

The Import menu contains the commands for import data from other CAD/CAM systems.




### Import ▸ DXF

Import polylines from DXF file and converts is to Shape Maker curves and approximation points representation.




Make Approximation Points - Import polyline points and convert it into approximation points.

Make lines - Create Shape Maker curves from DXF polylines.

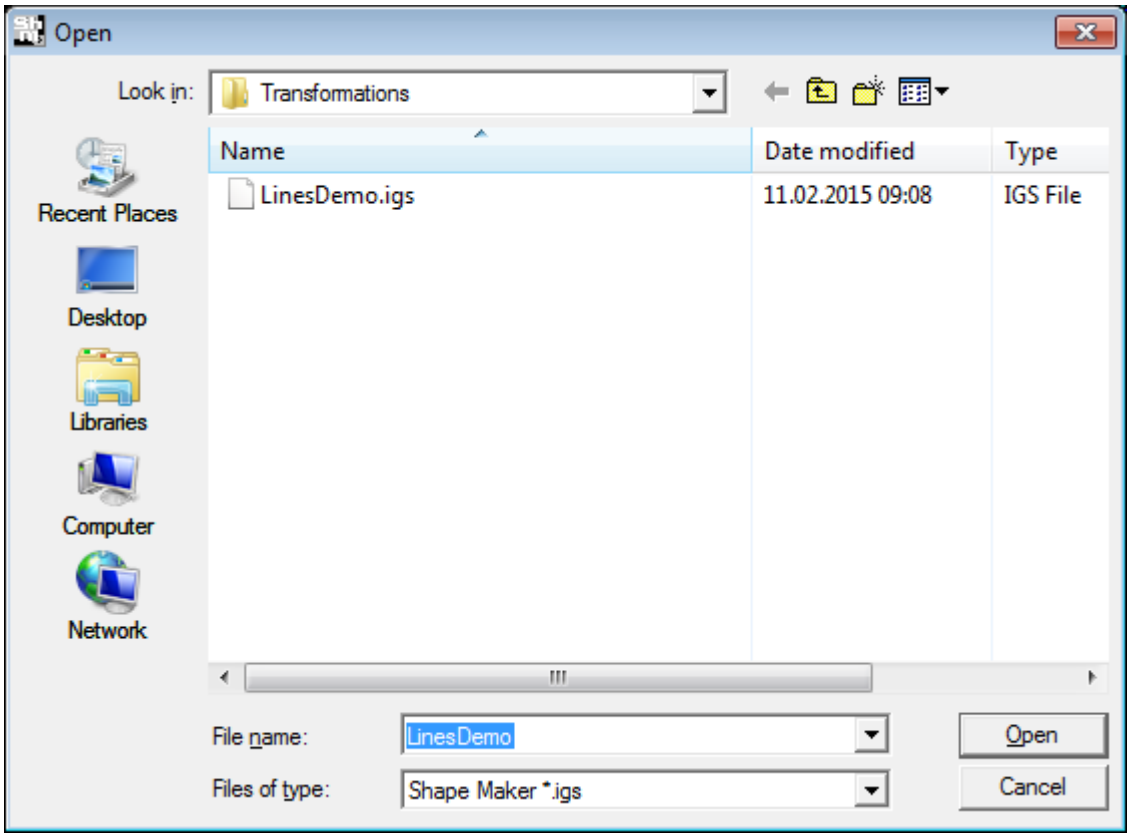
Interpolation precision (m) – maximum deviation of the Shape Maker curves from DXF polylines.

Detect knuckle angle (deg.) – minimum detected knuckle angle.

Maximal Step (m) – defines minimal length of line segment.

### Import ▸ IGES

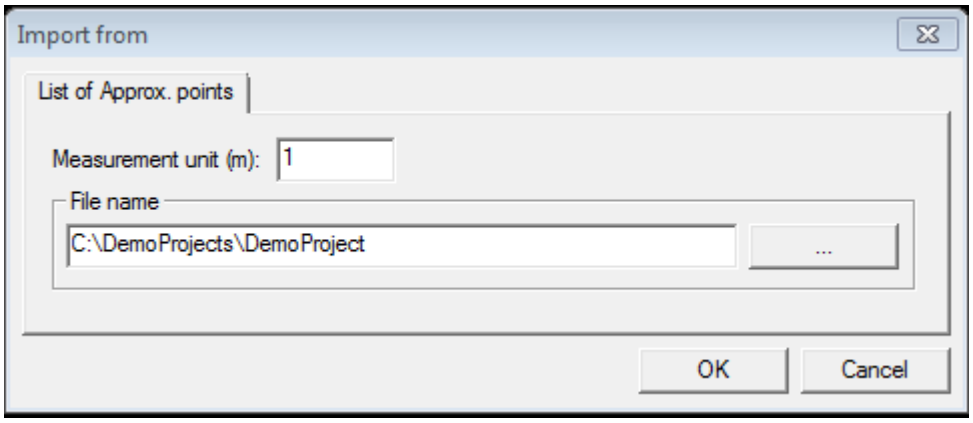
Import surfaces from IGES file and convert to Shape Maker surfaces.




All surfaces, points and lines will be imported into current block.

### Import ▸ Approximation points

This command read points from the LST file to the project file to use as approximation points.

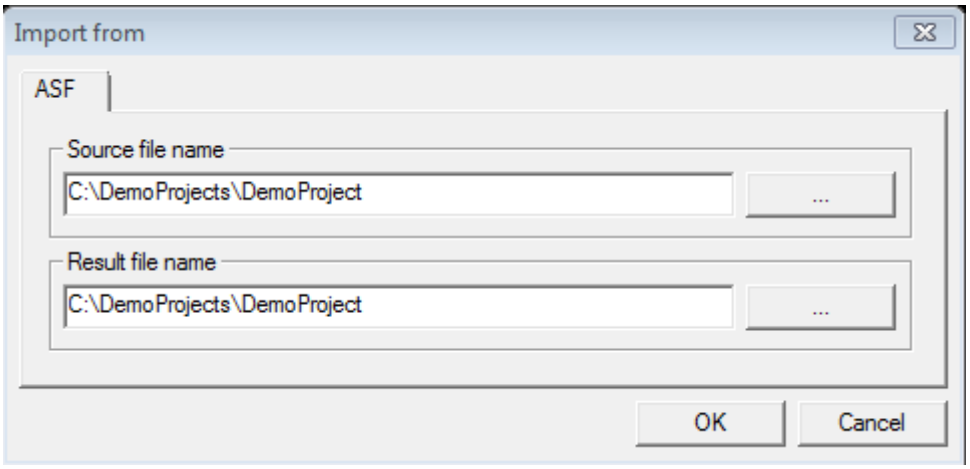



Measurement unit (m) - scale factor.

File name – name of the LST file

**Import ▸ ASF**

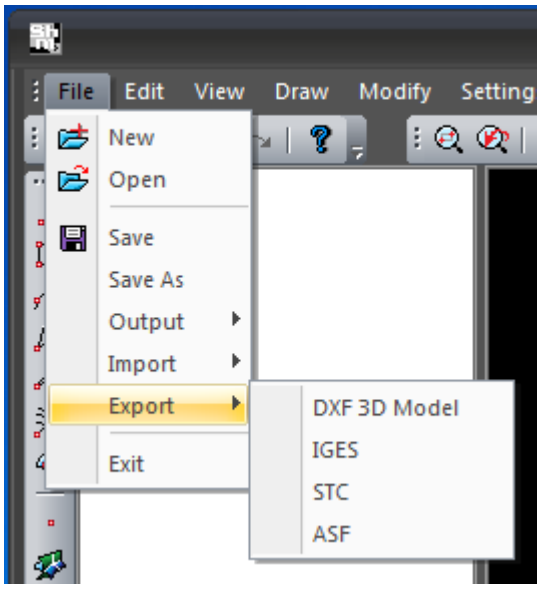
This command import the Shape Maker project previously saved in ASF file format.




Source file name – ASF file.

Result file name – Shape Maker SSL file where with import results.

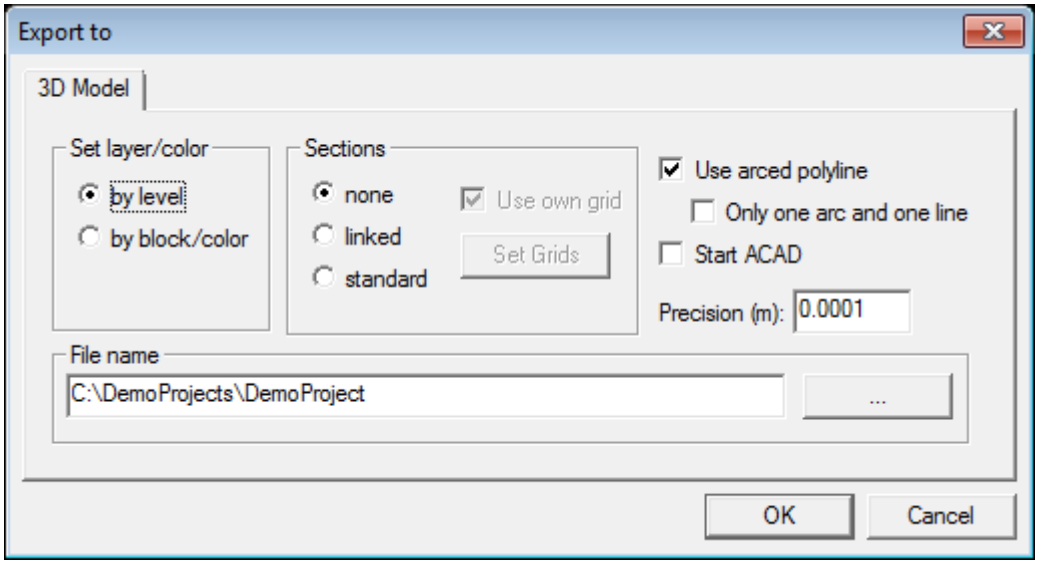
## File ▸ Export




### Export ▸ DXF 3D Model

Exports 3D project representation into DXF file.

Lines, surfaces and sections are transferred to the 3D model. The lines and the surface will be transferred in full, even if only a part of the line or surface is in the window.




Set layer/color: colors used when lines transferring.

by level - one color for all elements.

by block/color **-** the color of the project.

Sections: - draw sections lines.

none – no sections lines.

linked – draw section lines (one polyline per one section).

standard – draw section lines for each surface separately (one polyline per one surface patch).

Set Grids – definition of the own grid.

Use arced polyline **-** polylines with arcs for lines draw.

Only one arc and one line - Lines and arcs will follow one by one in polyline sequence.

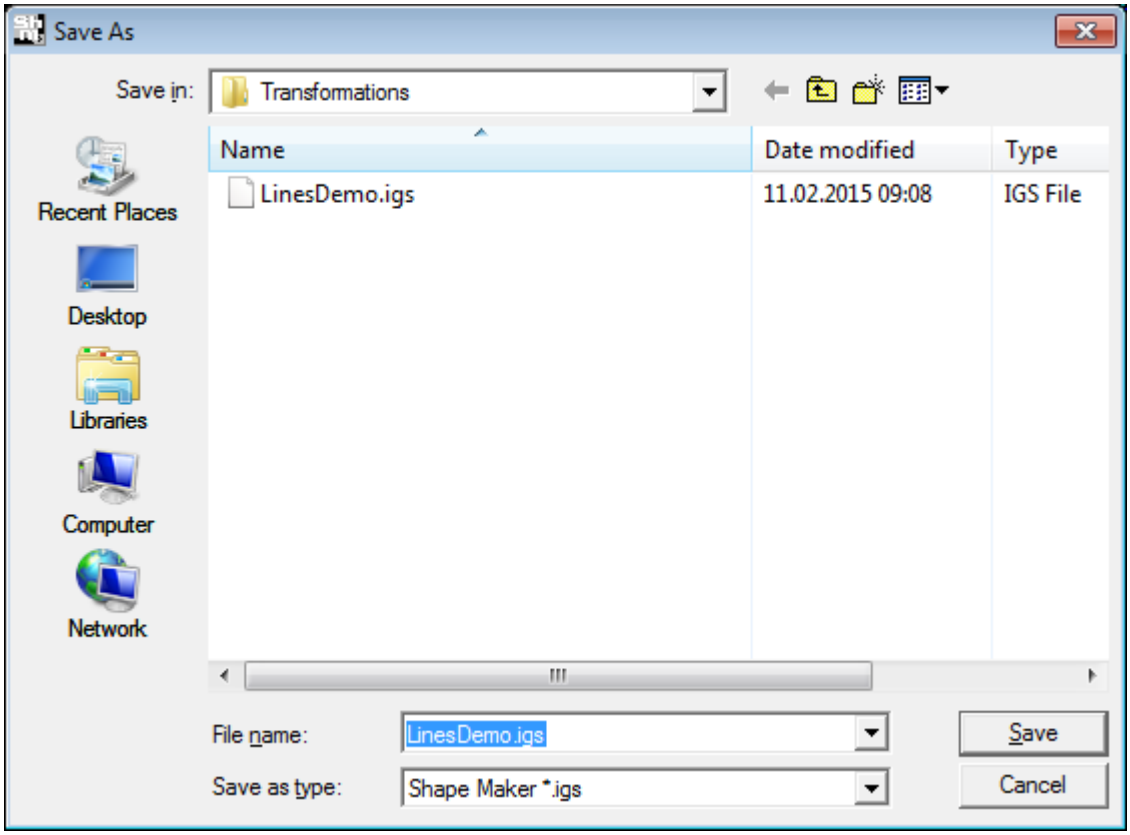
Start ACAD - loads the ACAD program after the plate is developed.

Precision (m) - precision for lines approximation in meters.

File name – the name of the output file.

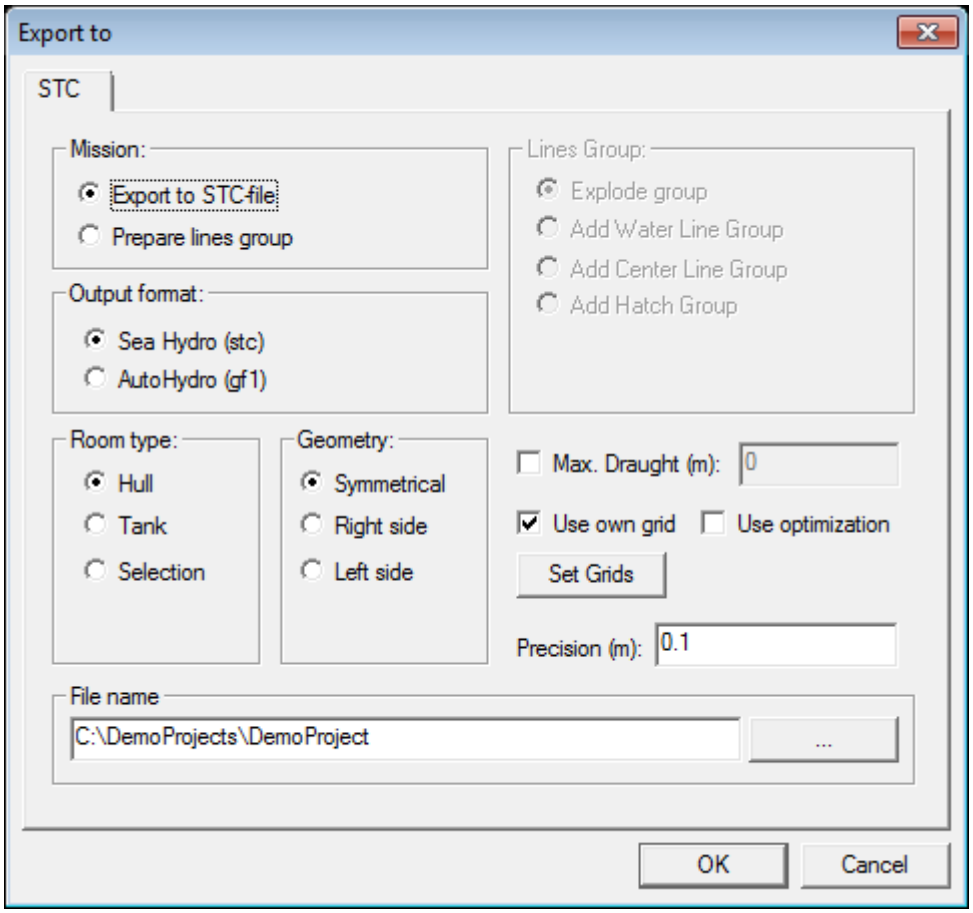
### Export ▸ IGES

Export to the IGES file format.



**Export ▸ STC**

Export to the STC file format for hydrostatic calculations.




Mission:

Export to STC-file – creating of STC file.

Prepare lines group – lines groups definition.

Lines Group:

Explode group – delete existing group

Add Water Line Group – standard margin of safety line definition.

Add Center Line Group – buttocks in CL definition.

Add Hatch Group – hatches and other openings line definition.

Output format:

Shape Maker– Shape Maker STC file

AutoHydro – gf1 file

Room kind:

Hull – hull.

Selection – compartment.

Tank – tank.

Type:

Symmetrical – symmetrical.

Right side - starboard.

Left side  - port side.

Max. Draught – maximum loaded draught

Use own Grid -  use none projects grid.

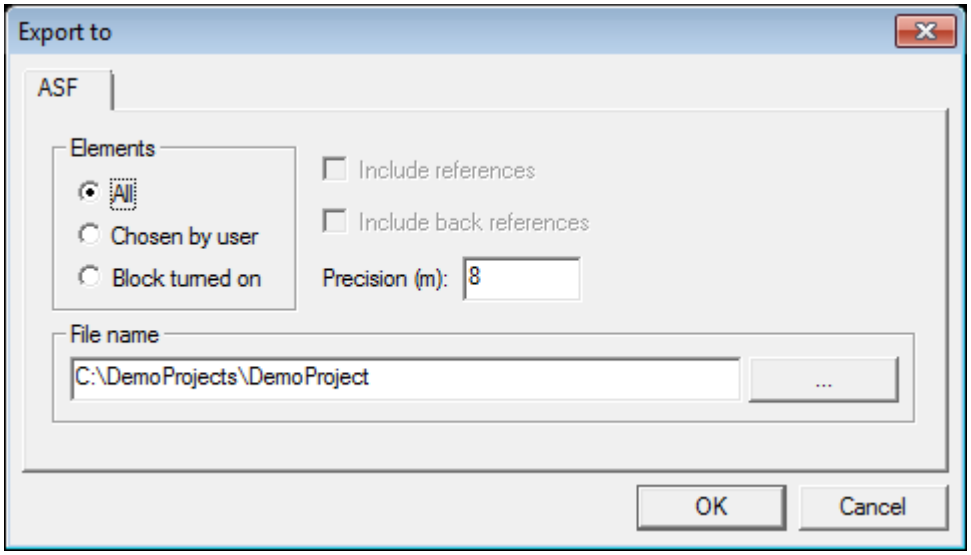
Use optimization -  optimize number of line points.

Set Grids - formation of the own grid.

Precision – precision of the sections representation.

**Export ▸ ASF**

Export to the ASF – project database text file dump.




Elements:

All – all elements of the project will be transferred.

Chosen by user – only specified elements will be transferred.

Block turned on – only visible elements will be transferred.

Include references **-** elements, which correspond to direct references, will be additionally transferred.

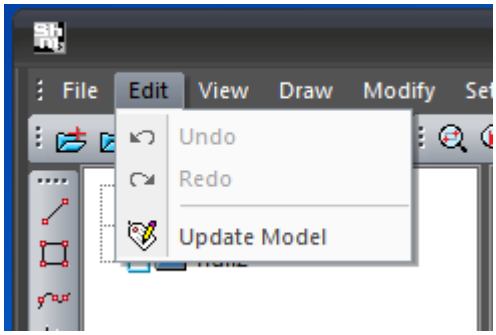
Include back references **-** elements, which correspond to back references will be additionally transferred.

Precision – precision of the text representation of the floating-point digits.

File name – name of the file where data will be stored.

# Edit

Menu Edit contains commands that allow undoing/redoing an operations and rebuilding the project elements.




## Edit ▸ Undo

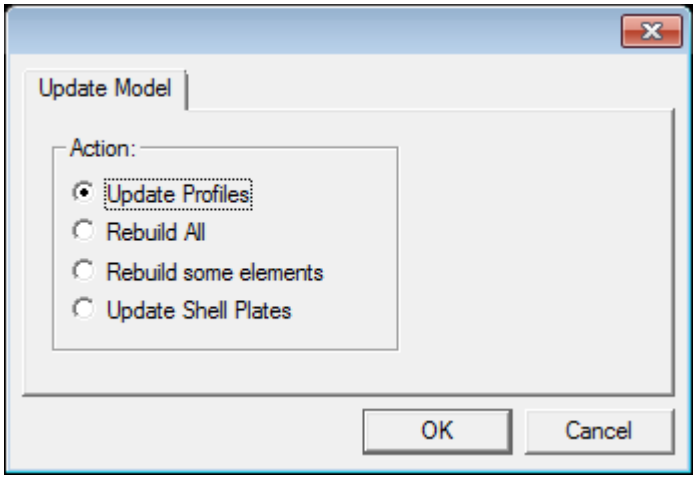
Undo command undoing previous edit operation.

## Edit ▸ Redo

Redo command redoing previous undo operation.

## Edit ▸ Update Model

Rebuild some elements of the project.




Action:

Update Profiles – all profile details will be rebuilt

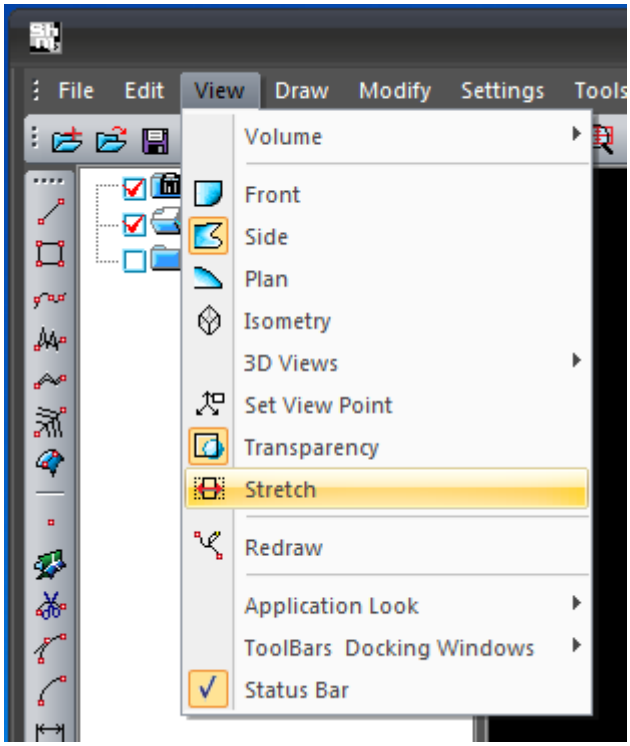
Rebuild All - all elements will be rebuilt

Rebuild some elements – only the element specified by the user and all other elements on which this element depends will be rebuilt.

Update Shell Plates – all plates of the project will be rebuilt.

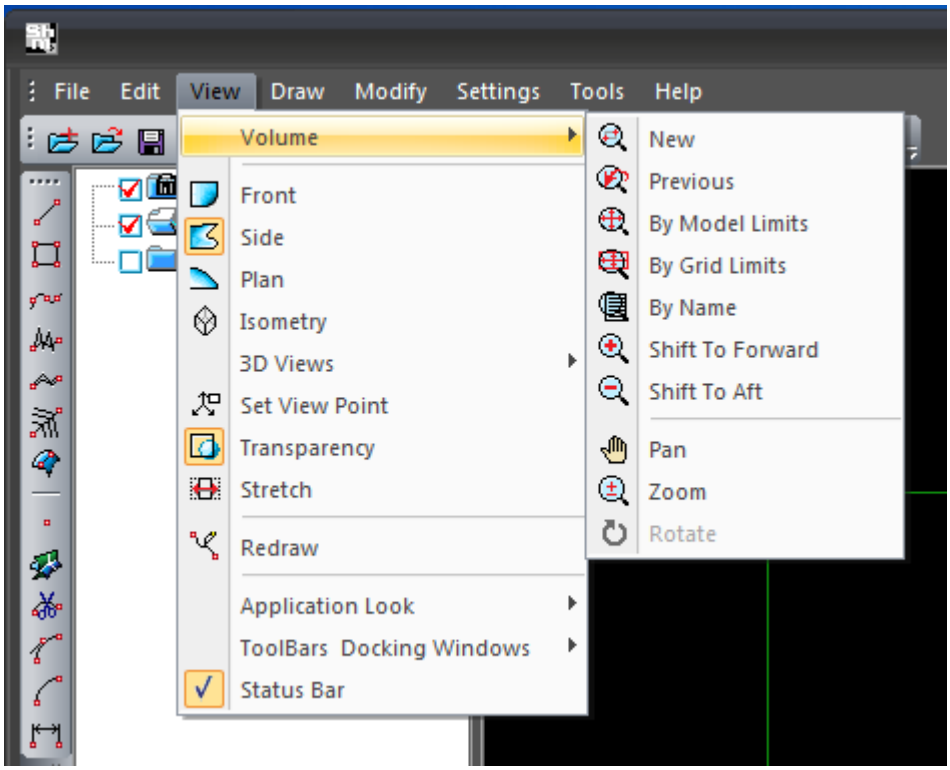
# View

View menu contains commands for visualization of the project and user interface managing tools.




## View ▸ Volume

The Volume menu commands are used to manage size and position of the current visible project volume.

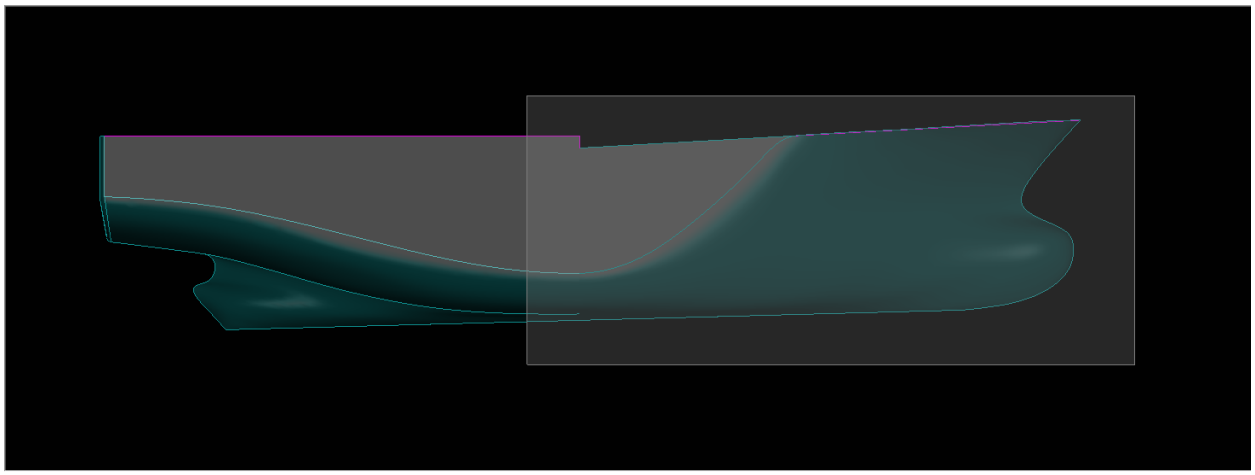


**Volume ▸ New**

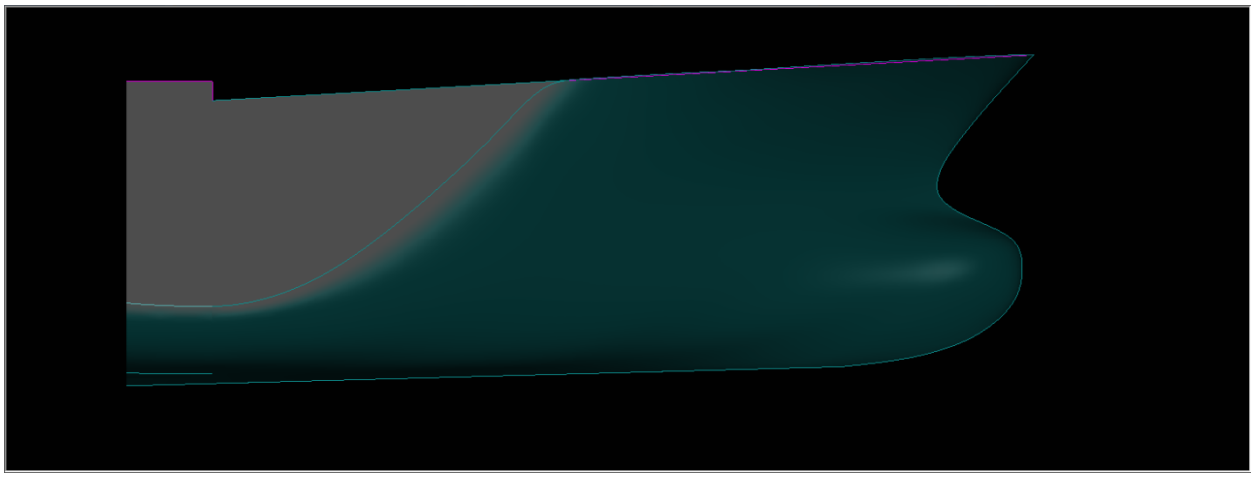
Set volume rectangle for the current projection.

Select Begin and end points of the rectangle to define new volume size and position on the current projection.

By request in the command line: "First corner: 0, 0, 0". Enter the coordinates or define position of the first point by the cursor. Move the marker, indicating boundaries of the visible window.

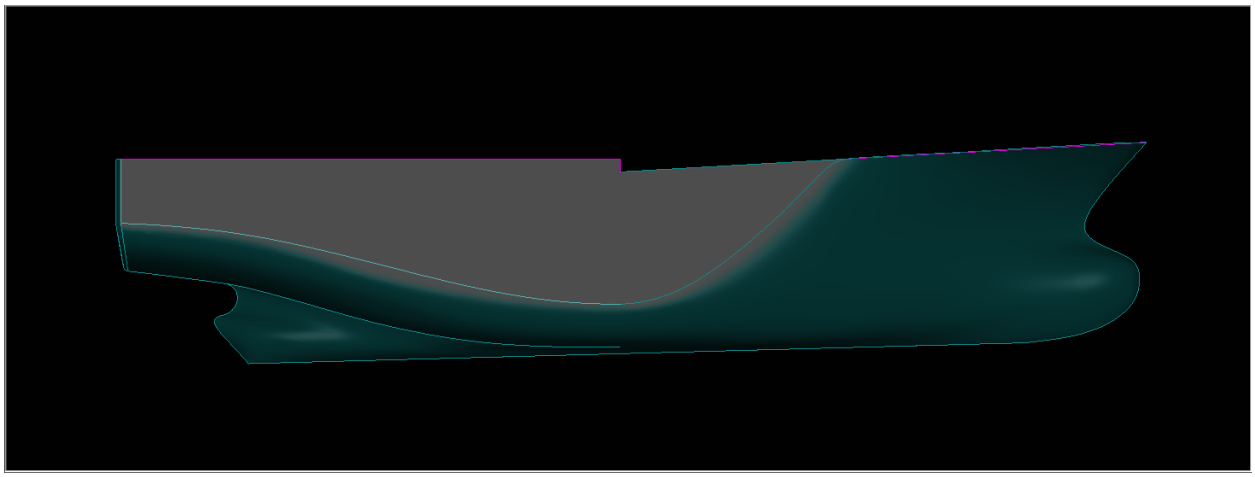


By request in the command line: "Second corner: 0, 0, 0". Enter the coordinates or define position of the second point by the cursor. An enlarged image of the model will appear on the screen.



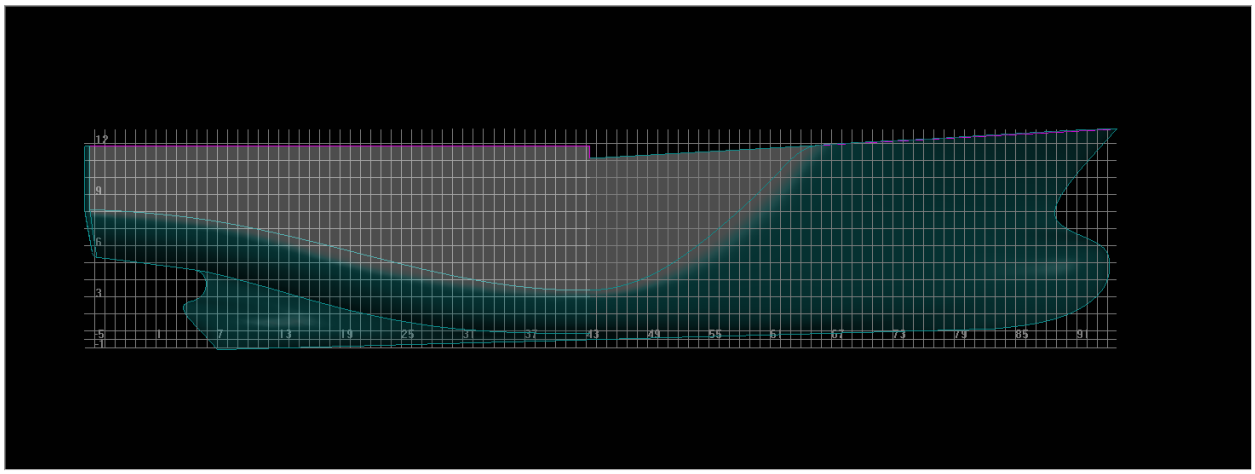
### Volume ▸ Previous

Return to previous project volume.



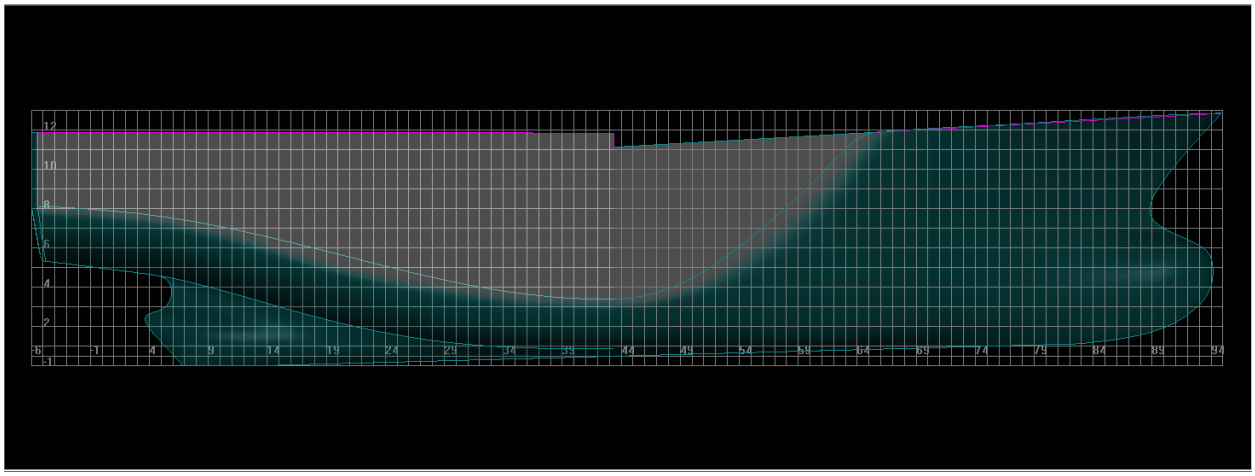
### Volume ▸ By Model Limits

Set up project volume by maximum visible model dimensions.



### Volume ▸ By Grid Limits

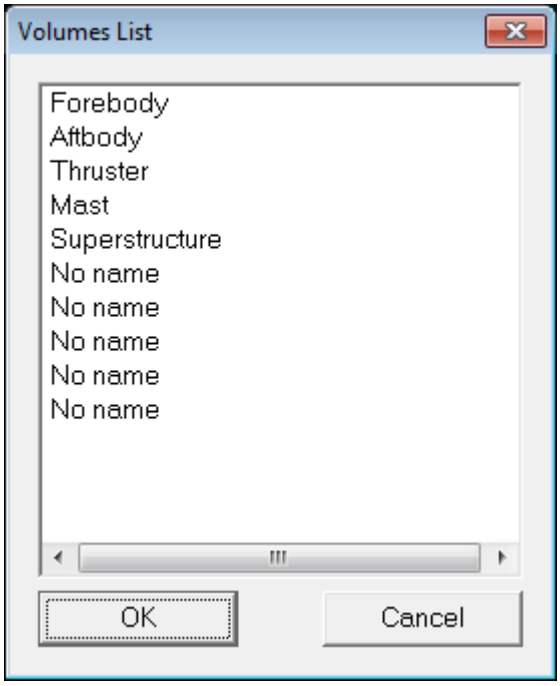
Set up project volume by project grid dimensions.



**Volume ▸ By Name**
This command allows saving 10 named volumes.

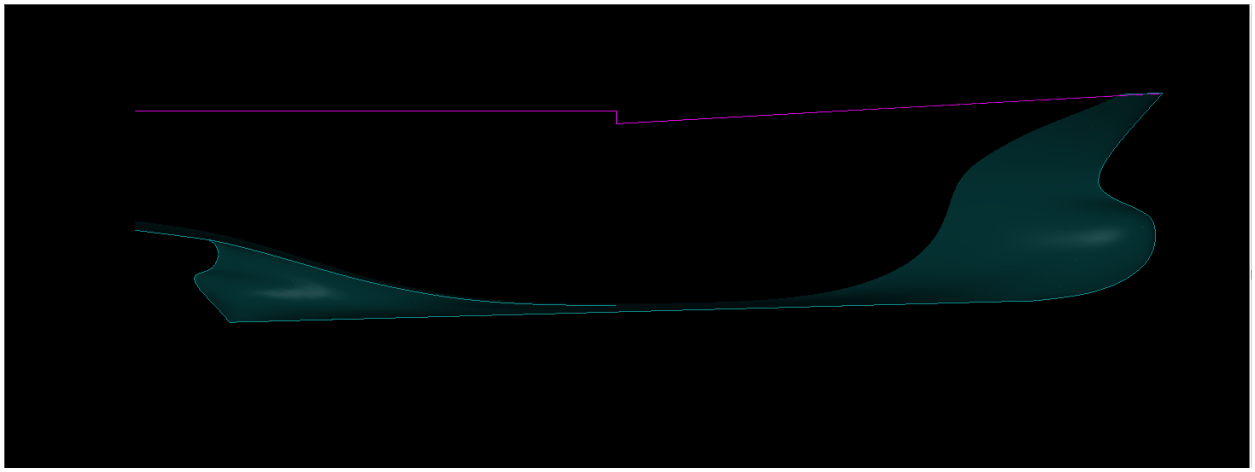
1. Set the required project volume.

2. Select the command **Volume ▸ By Name** from menu.
In the dialogue box enter name of the window to be saved.

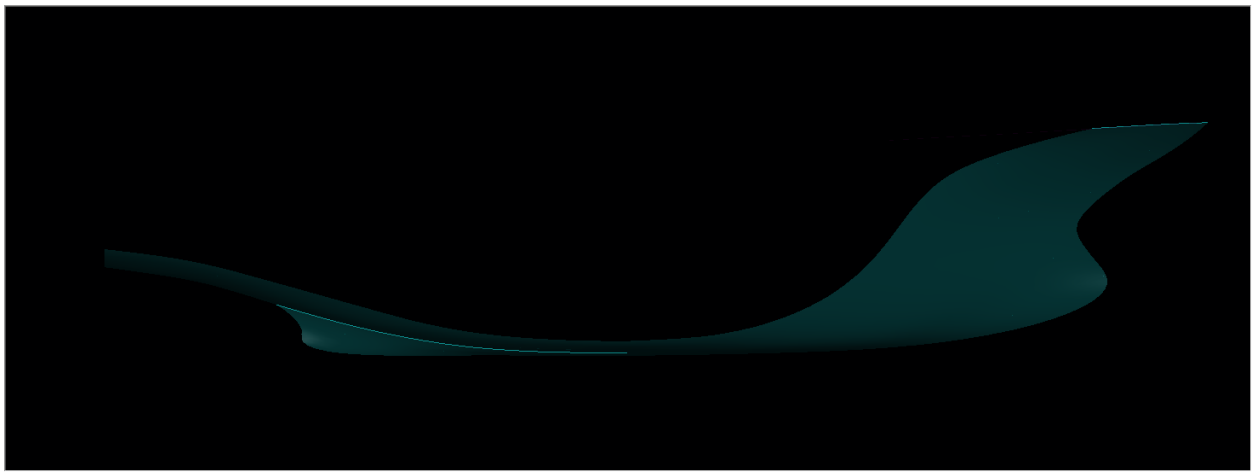



**Volume ▸ Shift to Forward**
Shift current volume in direction ”to user”.

Before:



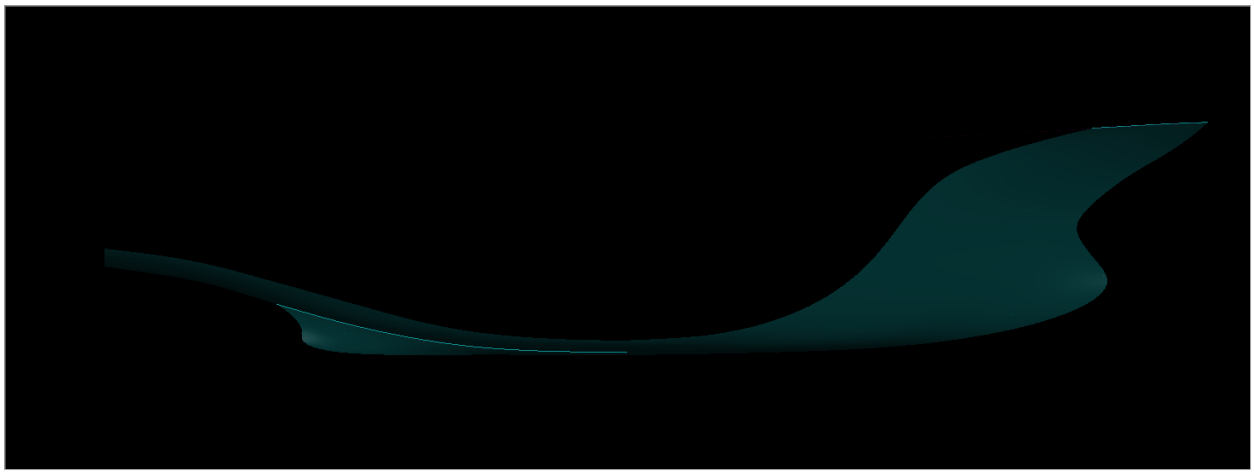
After:



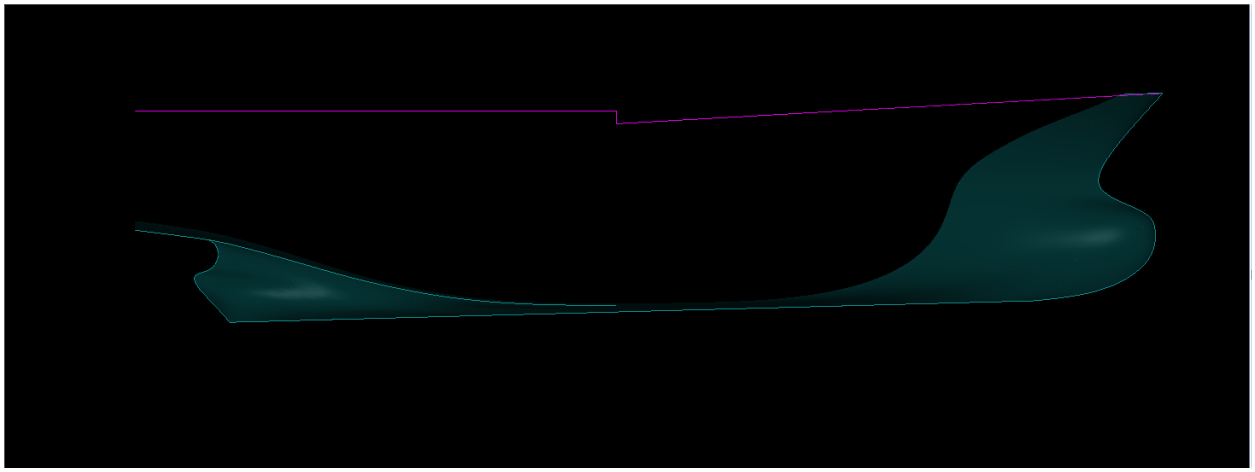
**Volume ▸ Shift to Aft**

Shift current volume in direction ”from user”.

Before:



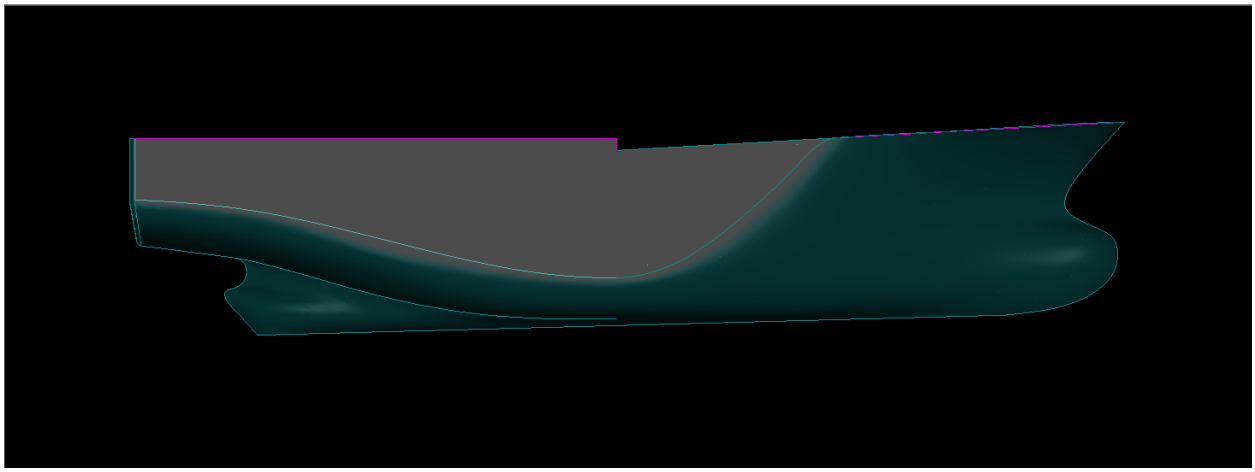
After:



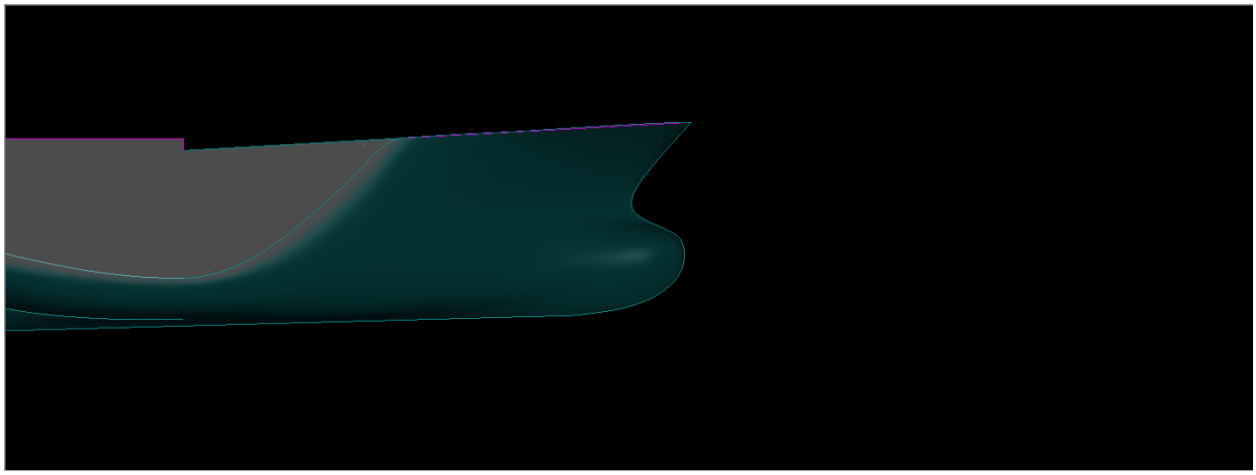
**Volume ▸ Pan**

Pan command is used for positioning visible objects in the window without changing current volume.

Before:



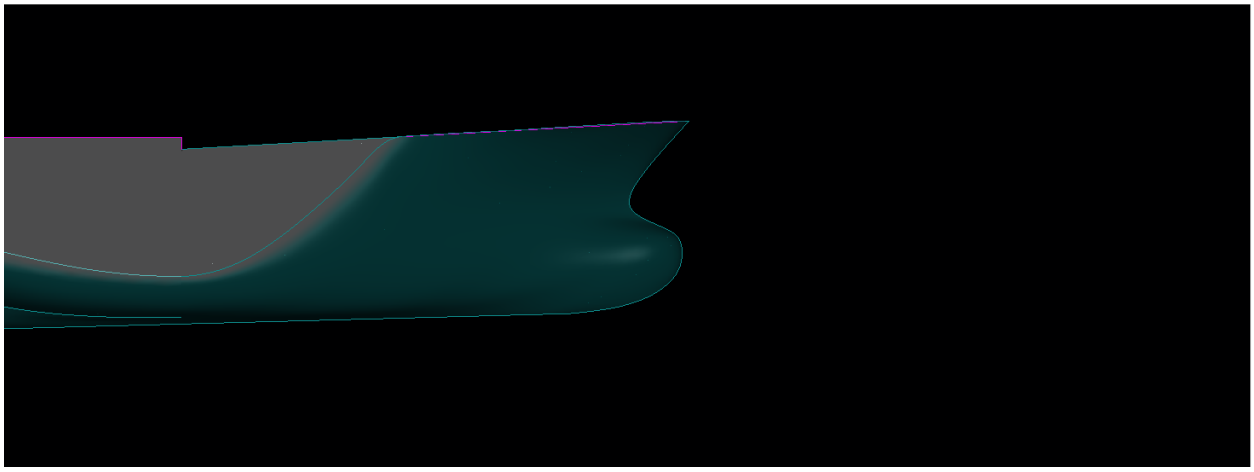
After:



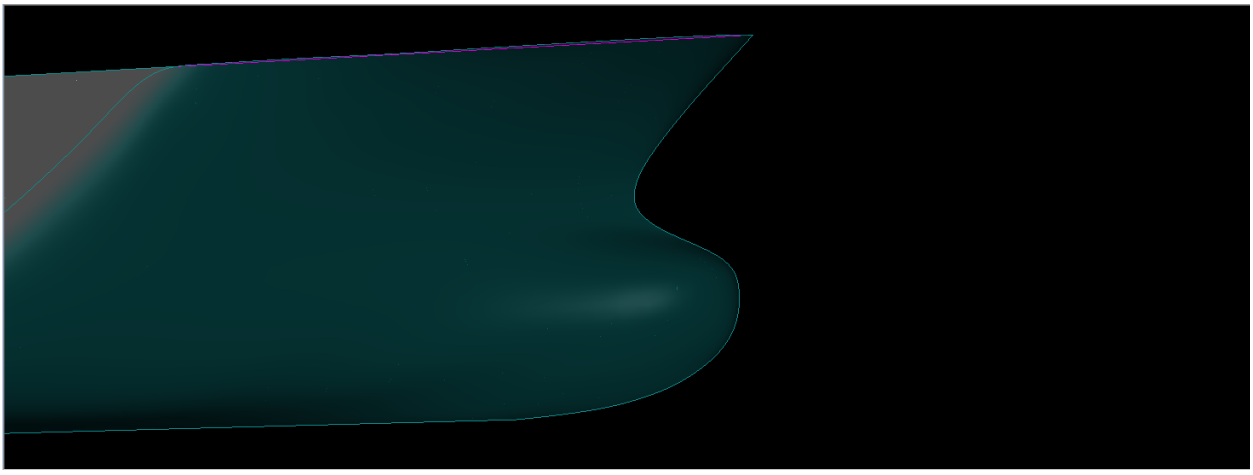
**Volume ▸ Zoom**

Zoom command is used for zooming in/out visible objects in current window.

Before:



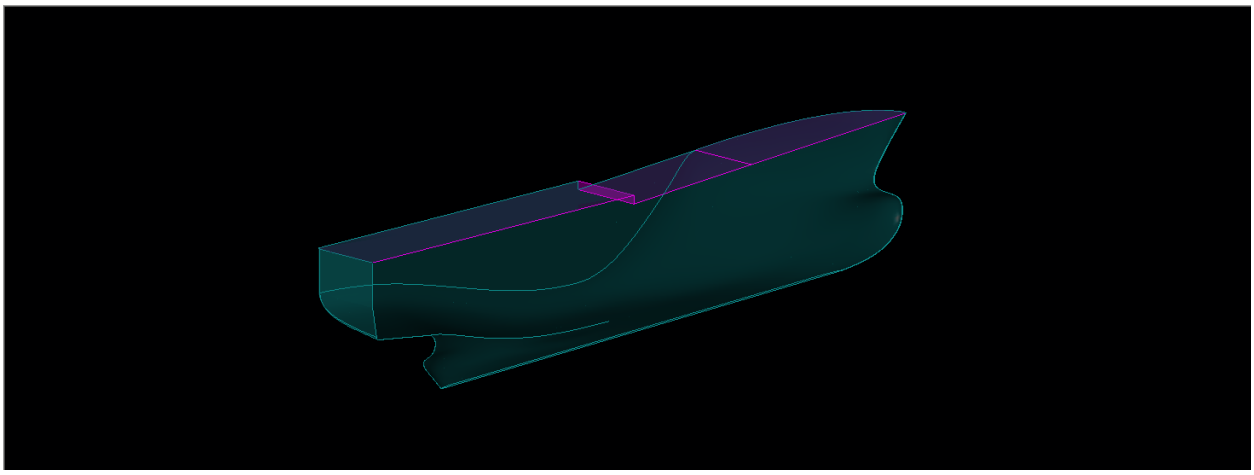
After:



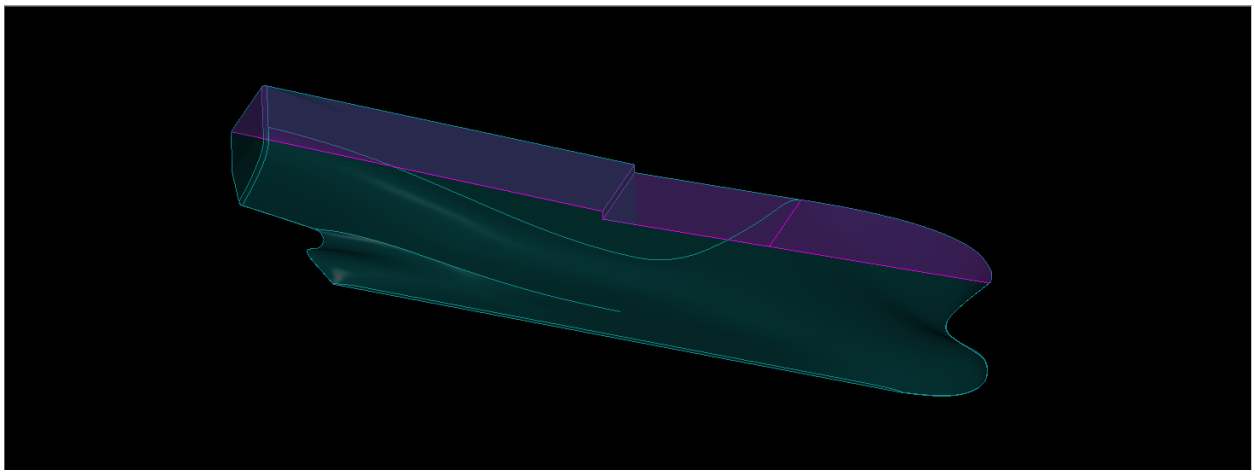
**Volume ▸ Rotate**

Rotate command is used for rotating current volume in the window. Available only in 3D view.

Before:

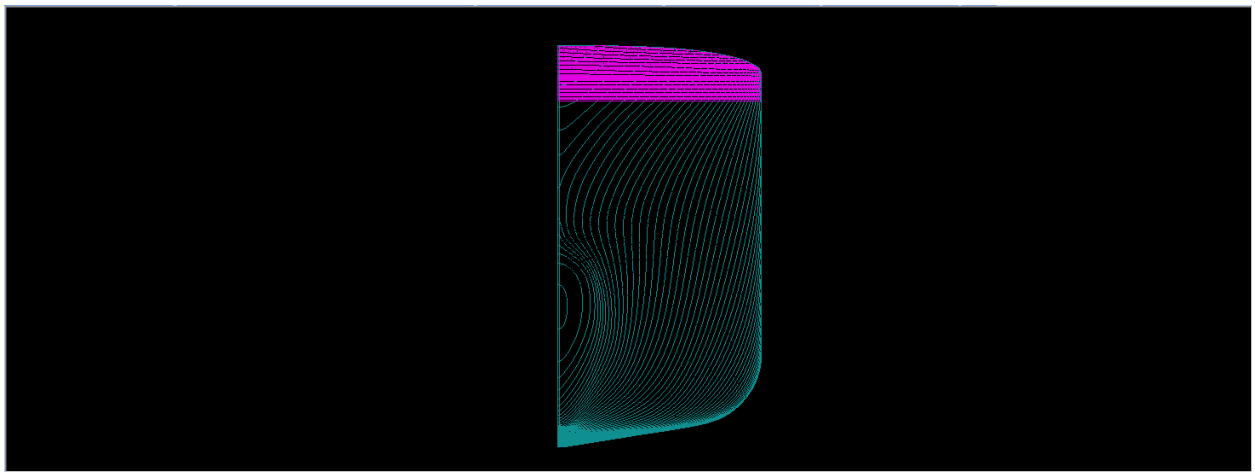


After:



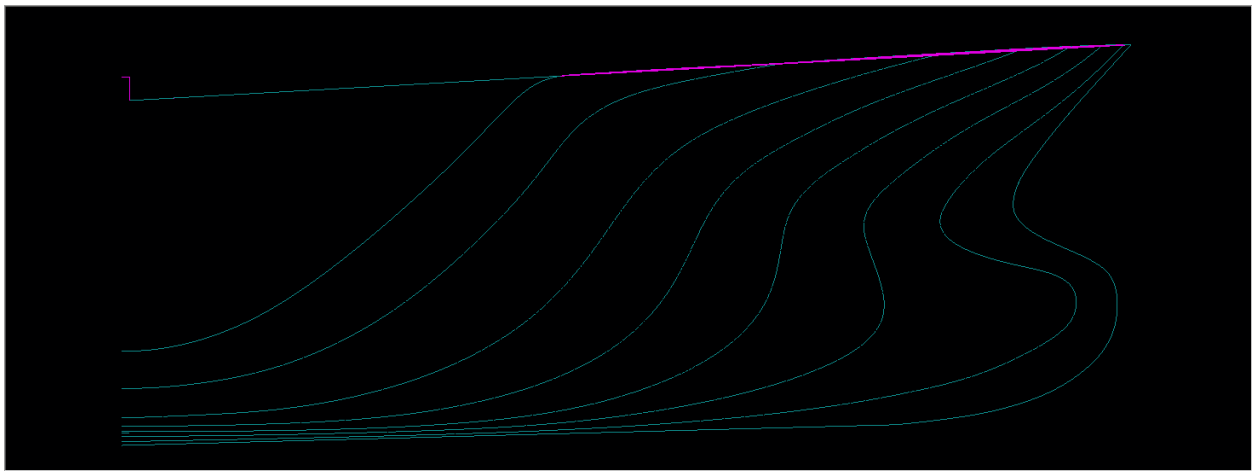
## View ▸ Front

Front command is used for setting the current view Front.



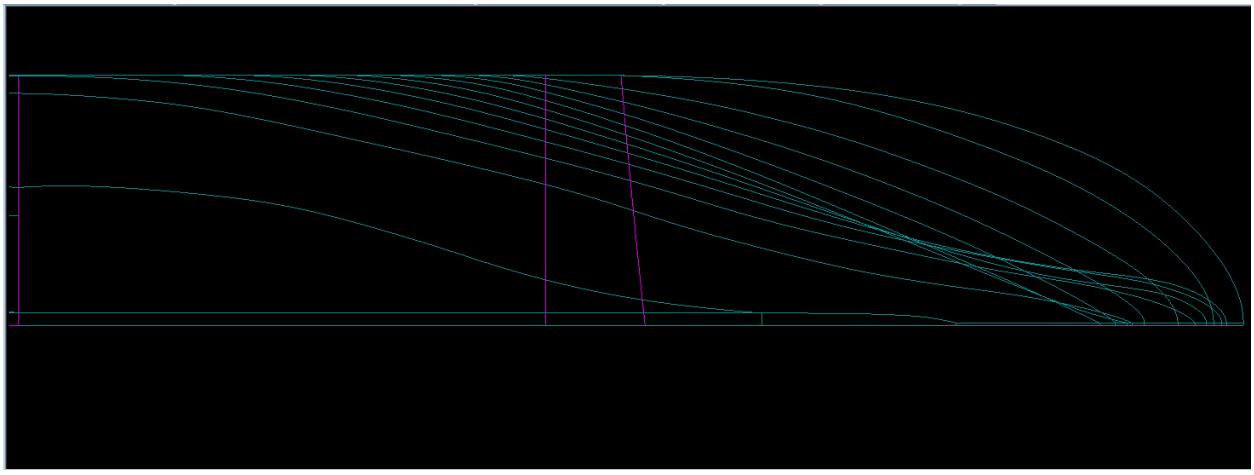
## View ▸ Side

Side command is used for setting the current view Side.



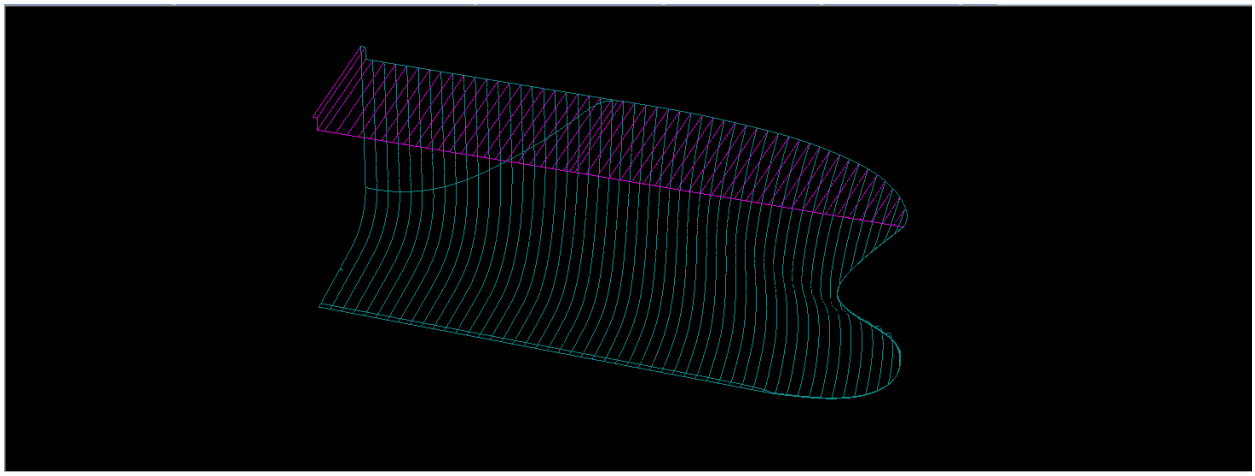
## View ▸ Plan

Plan command is used for setting the current view Plan.



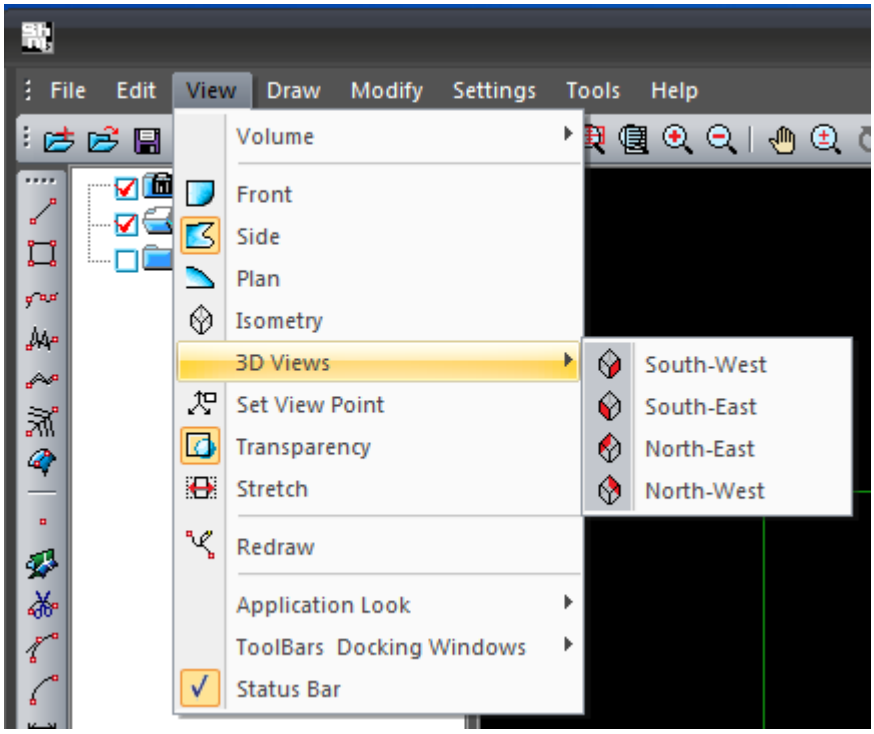
## View ▸ Isometry

Isometry command is used for setting the current view as 3D view.



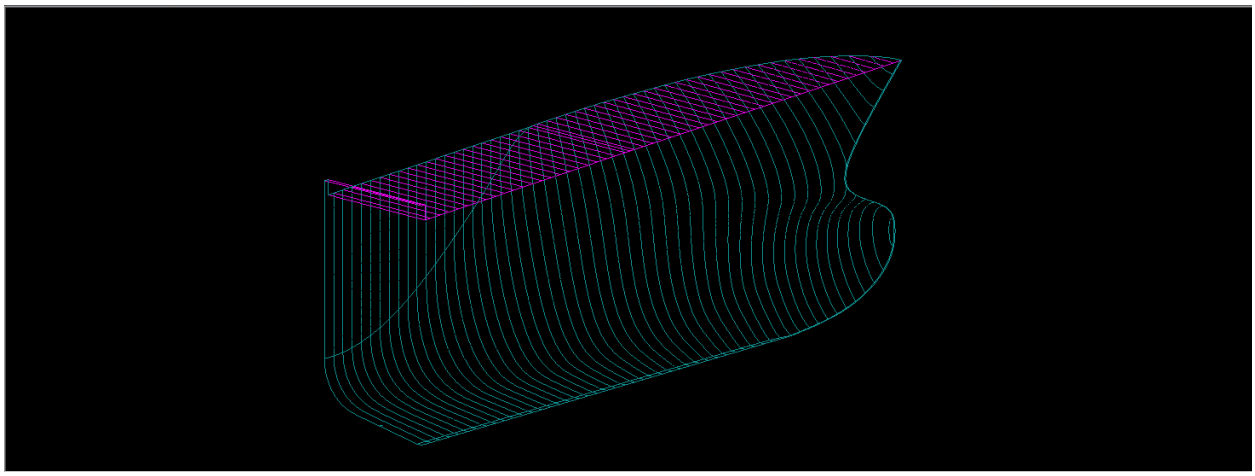
## View ▸ 3D Views

3D Views menu commands are used to set 4 standard Viewpoints.



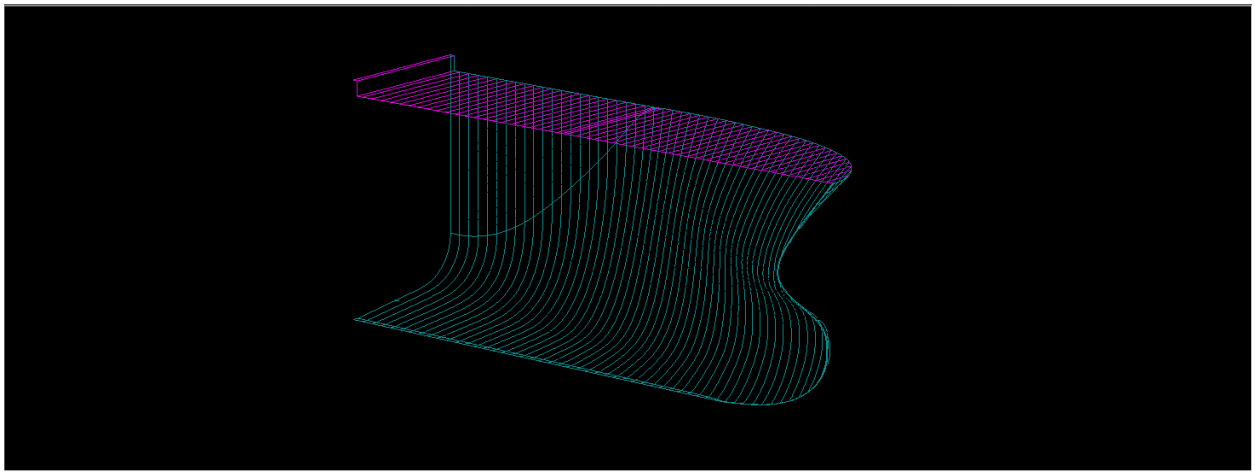
### 3D Views ▸ South-West

This command set up the current point of view as South-West.



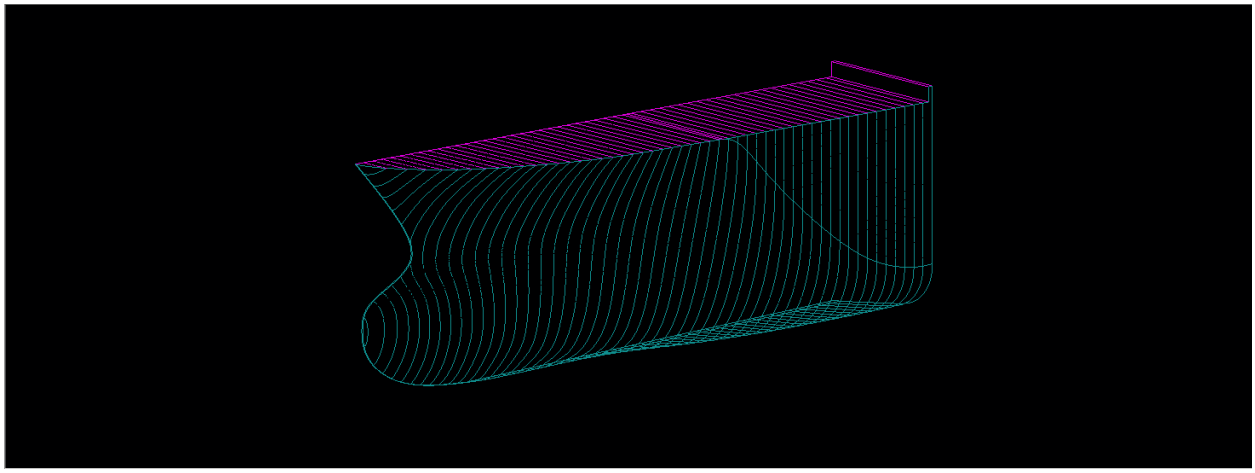
### 3D Views ▸ South-East

This command set up the current point of view as South-East.



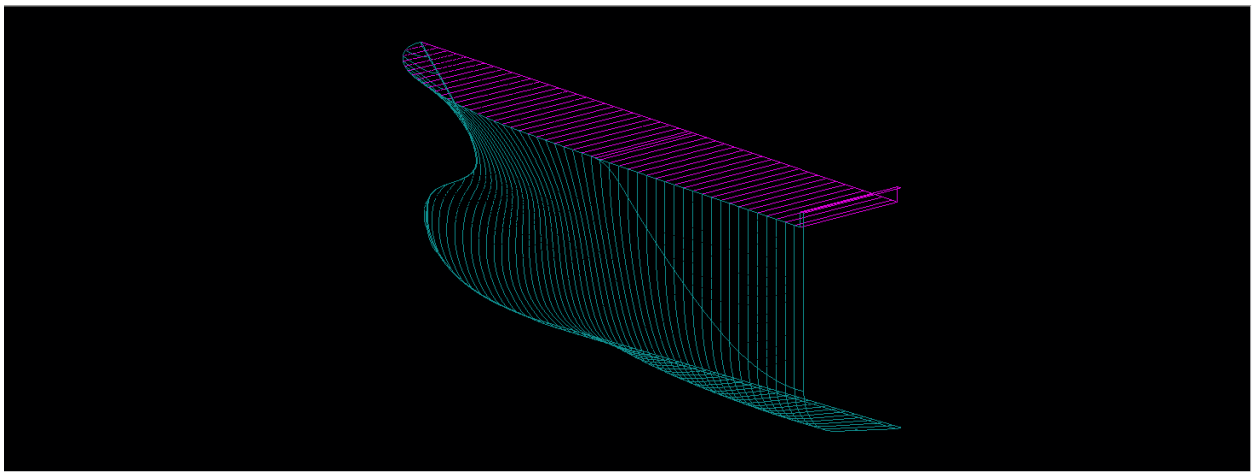
### 3D Views ▸ North-East

This command set up the current point of view as North-East.



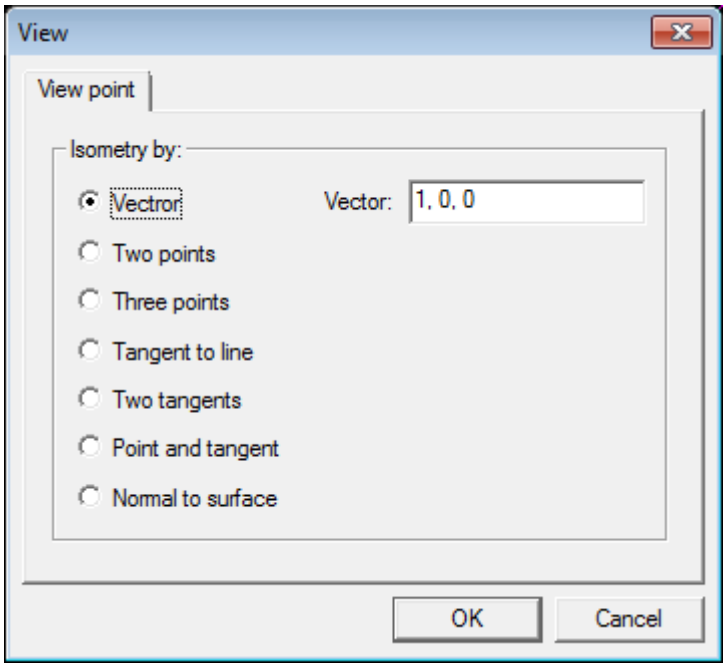
**3D Views ▸ North-West**

This command set up the current point of view as North-West.



## View ▸ Set View Point

This command set up user defines view point for 3D view.




The View Point can be set up by the following methods:

Vector – arbitrary vector.

Select Vector from the list. Input vector coordinates into the Vector field, then click OK .

Two points – vector determined by two points.

Select Two points from the list. Click OK. Input first and second points by cursor or coordinates.

Three points – vector as normal to plane determinate by three points.

Select Three points from list. Click OK. Input first, second and third points by cursor or coordinates.

Tangent to line- vector as tangent to line.

Select Tangent to line from list. Click OK. Select by cursor line for tangent determination.

Two tangents – vector as normal to plane, parallel with two tangents.

Select Two tangents from list. Click OK. Select by cursor lines for tangents determination.

Point and tangent - normal to plane determined by point and tangent to line.

Select Point and tangent from list. Click OK. Input point by cursor or coordinates. Select by cursor line for tangent determination.

Normal to surface – vector normal to surface.

Select Normal to surface from list. Click OK. Select by cursor surface for tangent determination.

## View ▸ Transparency

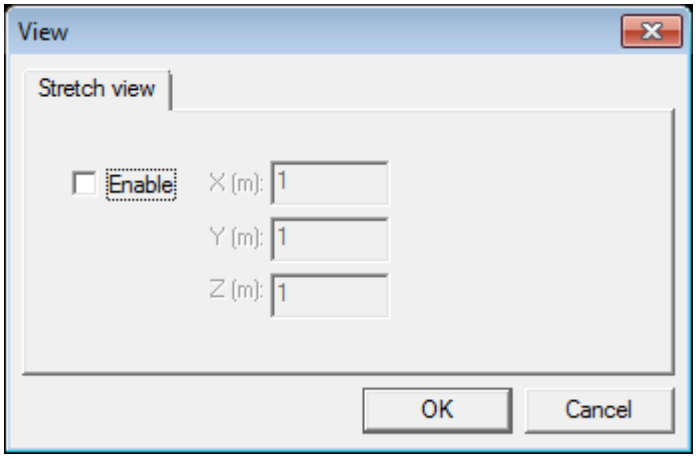
This command switches transparency of the surfaces.

Transparent surfaces:

Not transparent surfaces:

## View ▸ Stretch

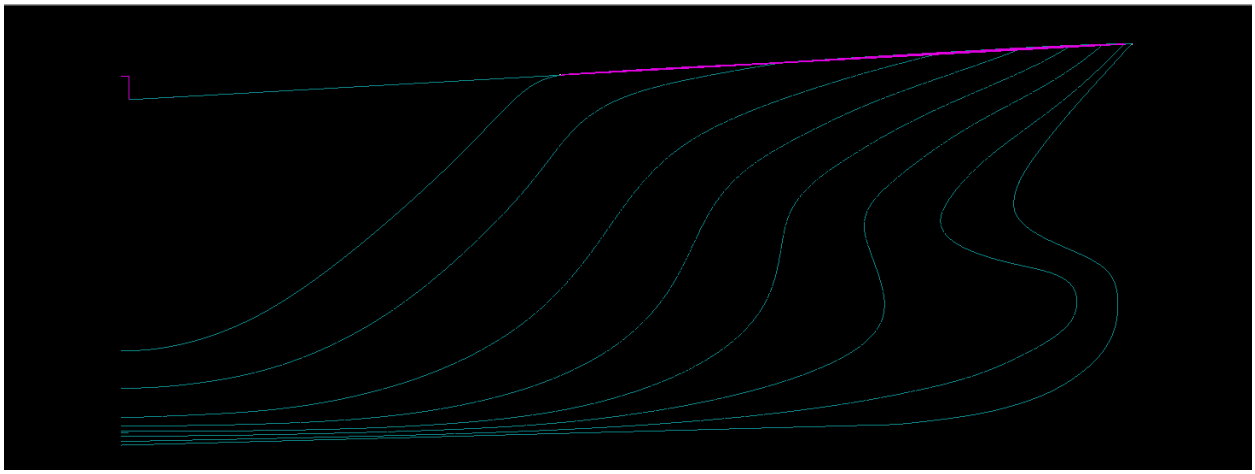
This command set up scale factors in axes X, Y, Z and performs transformation of the image. This transformation affects only visual representation of the model.



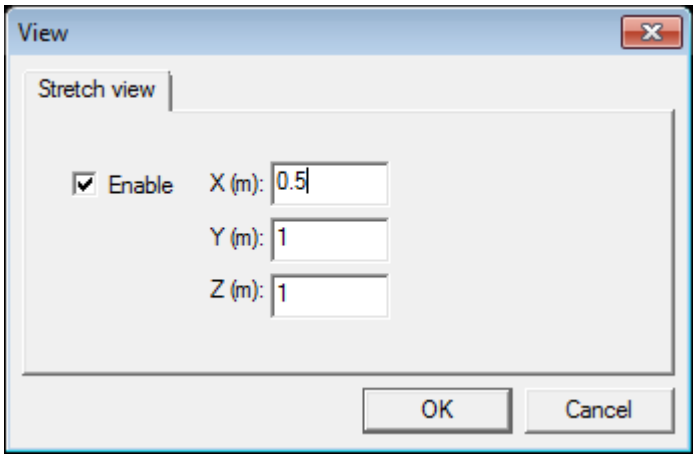

Enable **-** enables the transformation.

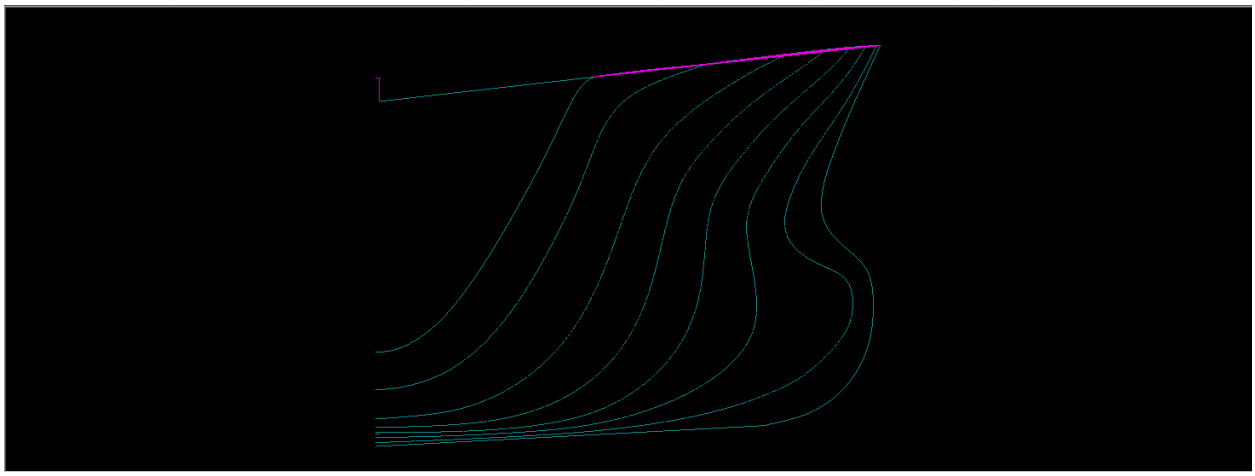
Enter the scale factors in axes X, Y, Z. An image in accordance with the entered factors will appear on the screen.

Before:



After:



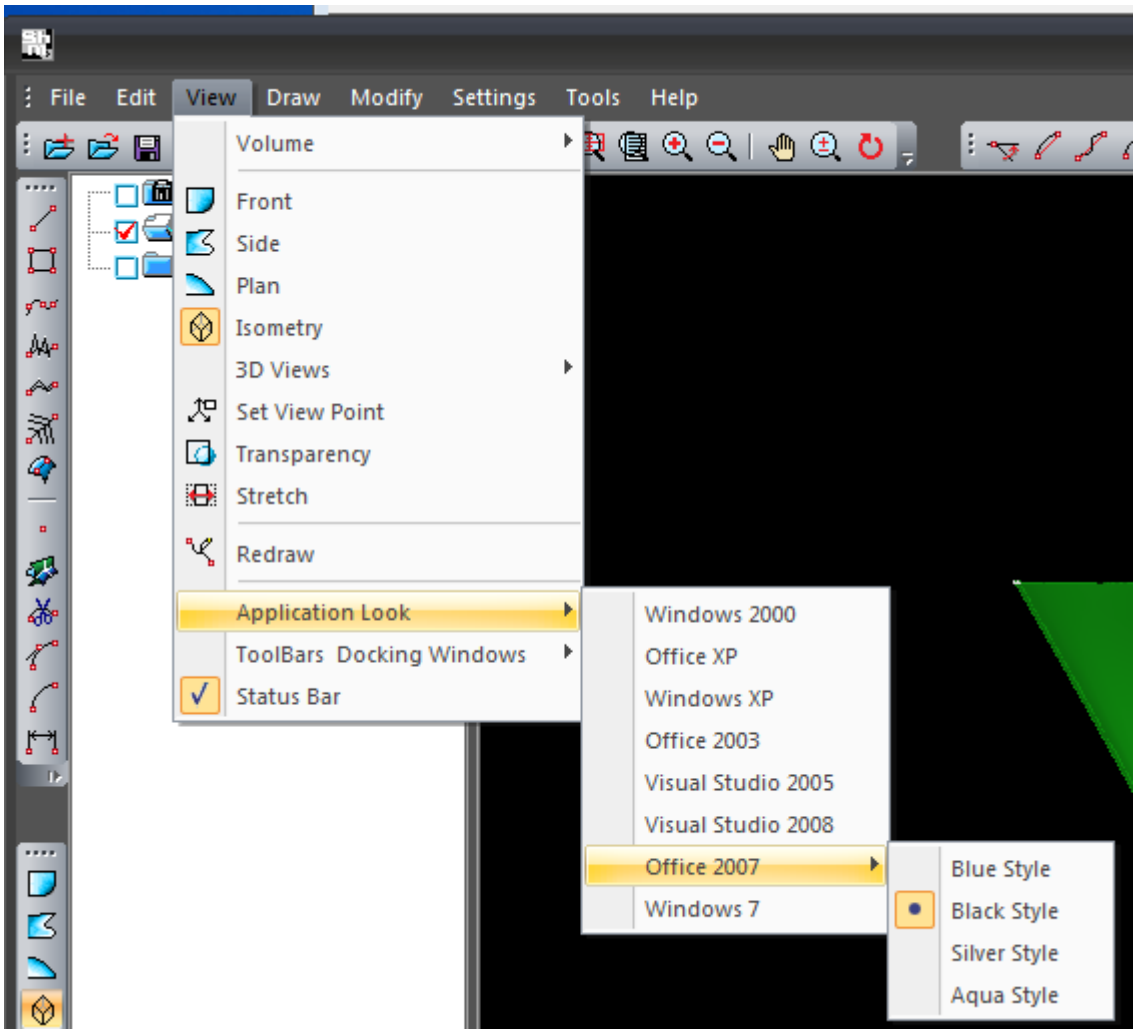



## View ▸ Redraw

This command redraws images of all visible elements.

## View ▸ Application Look

By this command user can choose different looks of the program.




## View ▸ Toolbar Docking Windows

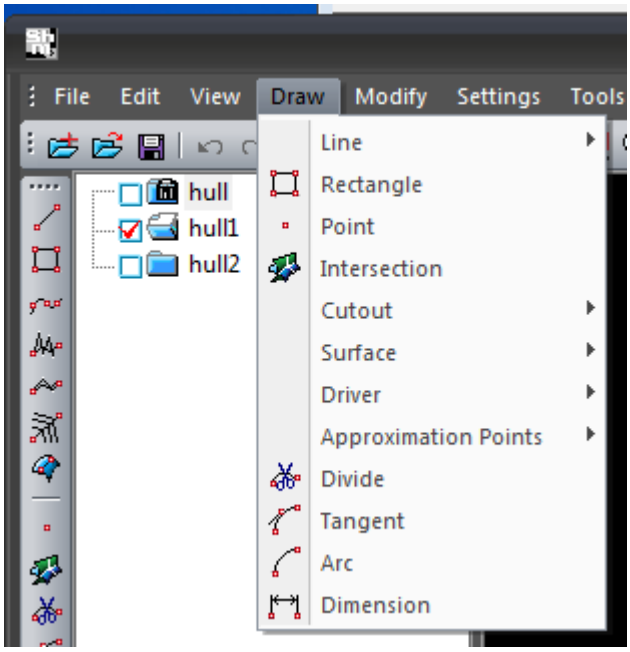
By this command user having access to Windows standard toolbars docking options.

## View ▸ Status Bar

This command switch on/off status bar visibility.

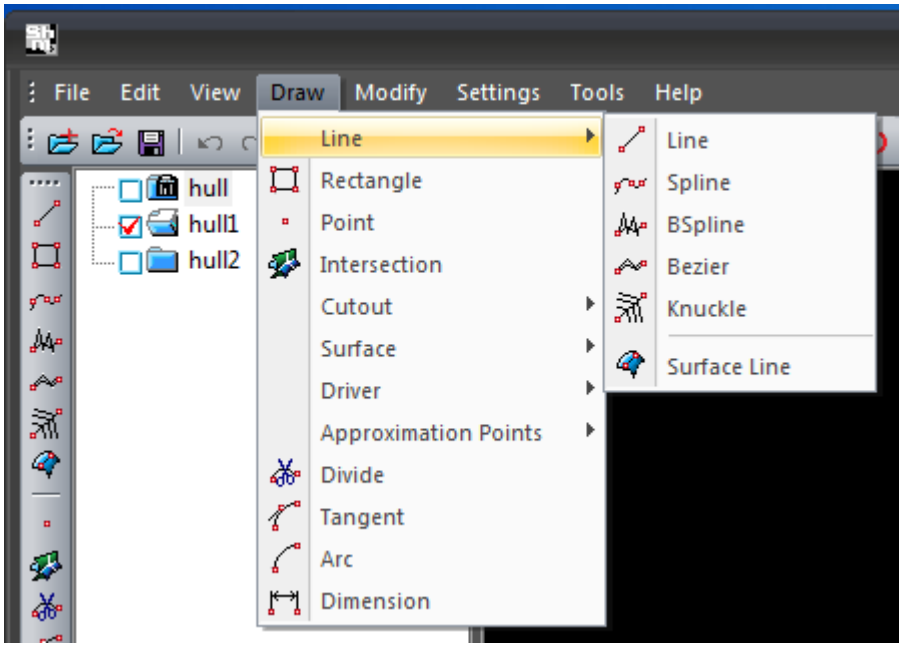
# Draw

Draw menu contains commands for input new graphical elements into the project model.




## Draw ▸ Line

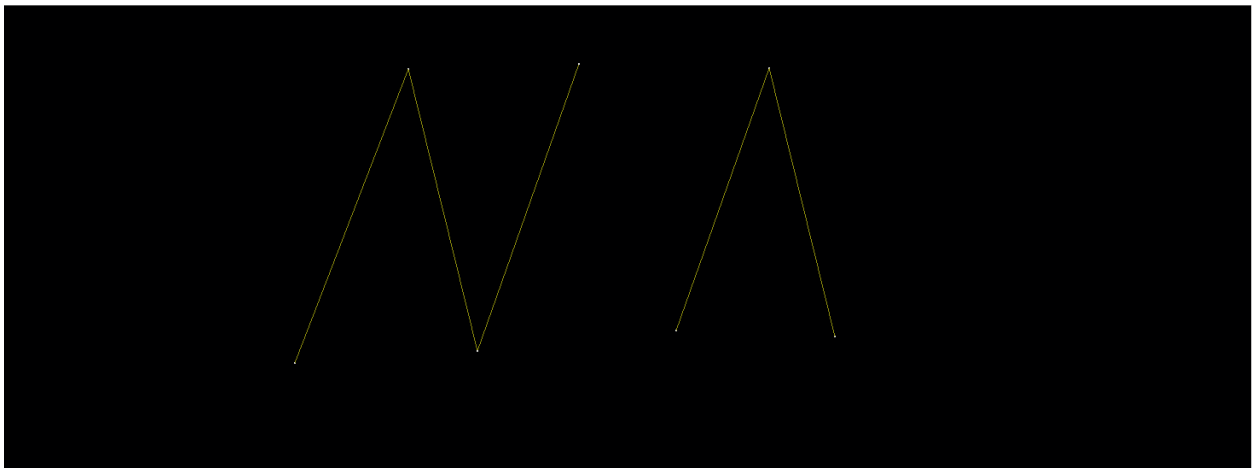
This command is used for input any type of lines into the project model.



### Line ▸ Line

Draw straight line between two points.

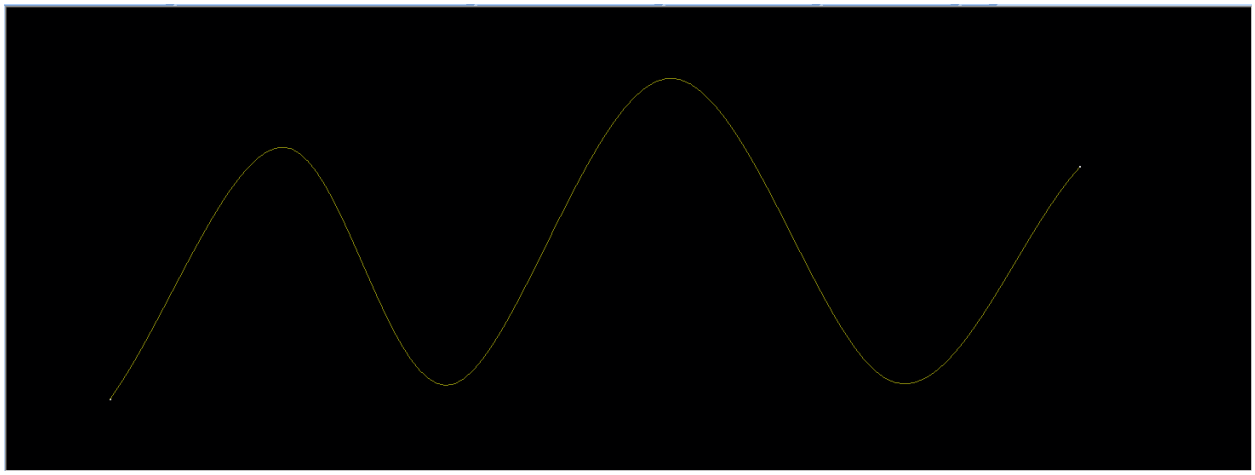
Input first point of the line by cursor or coordinates. Input second point. New line will appear in the model. Input new second point - new line will appear. And so on. To stop input lines chain press Esc button or right mouse button. To stop lines input press Esc button or right mouse button one more time.



### Line ▸ Spline

Draw spline by the set of points.

Input first point of the line by cursor or coordinates. Input second point. New line will appear in the model. Input next point. Resulting line will pass all inputted points. To stop input spline press Esc button or right mouse button. To stop splines input press Esc button or right mouse button one more time.

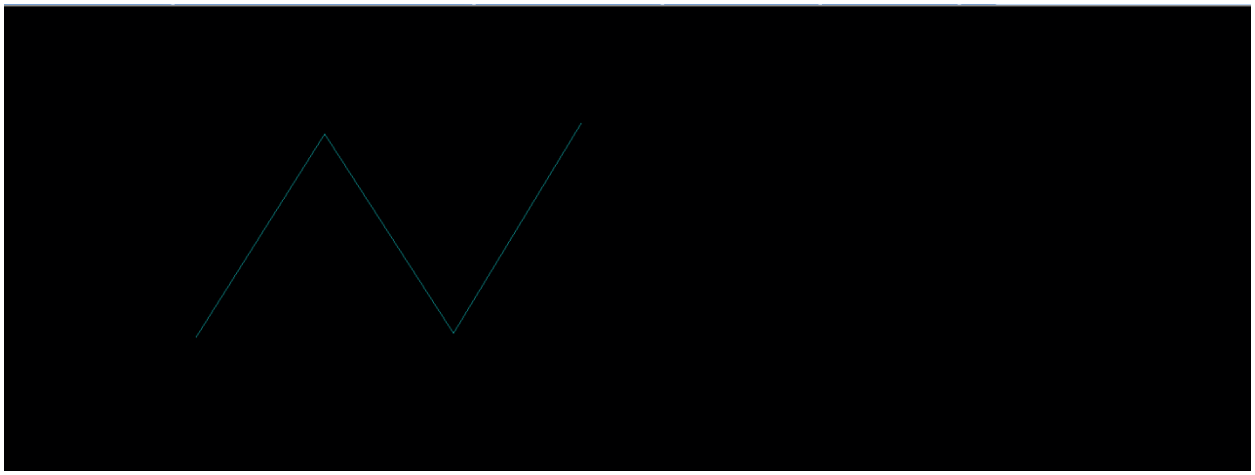


**Line ▸ B-Spline**

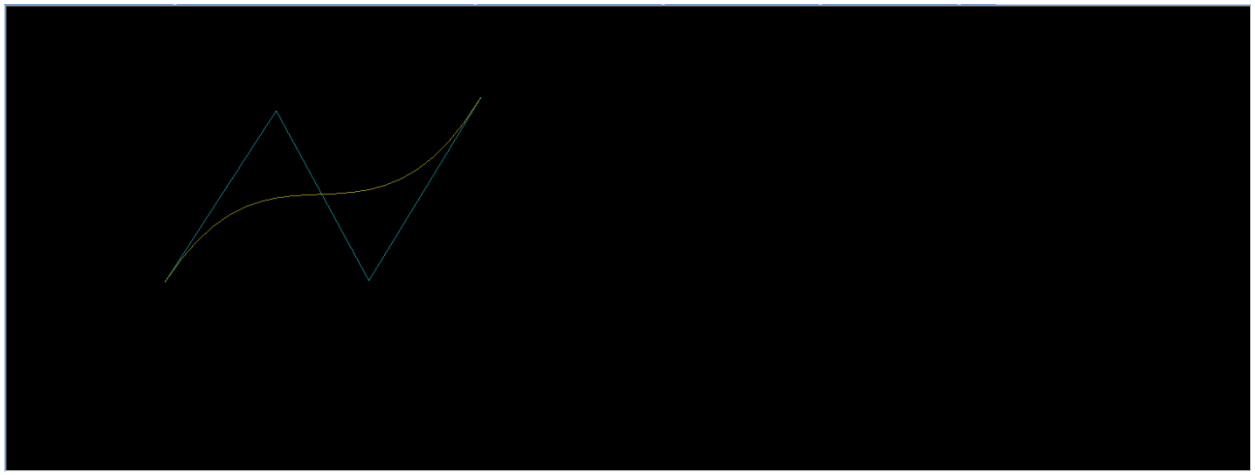
Draw b-spline by the set of points. To draw b-spline curve by control polygon minimum four points required.

Input first point of the line by cursor or coordinates. Input tree more points. New b-spline will appear in the model. Resulting line will follow control polygon of b-spline. To stop input b-spline press Esc button or right mouse button. To stop b-splines input press Esc button or right mouse button one more time.

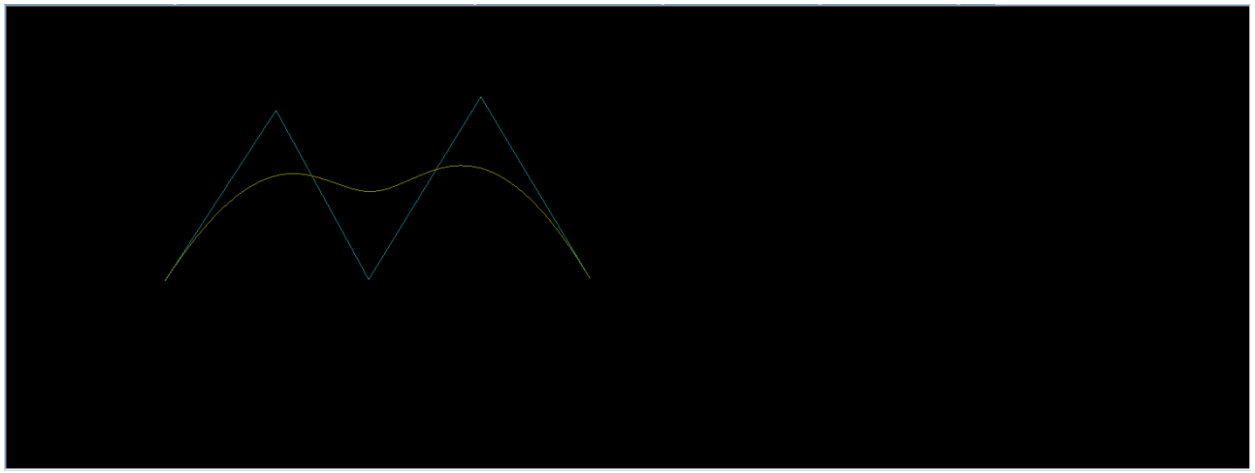
Minimal control polygon:



Control polygon and b-spline:



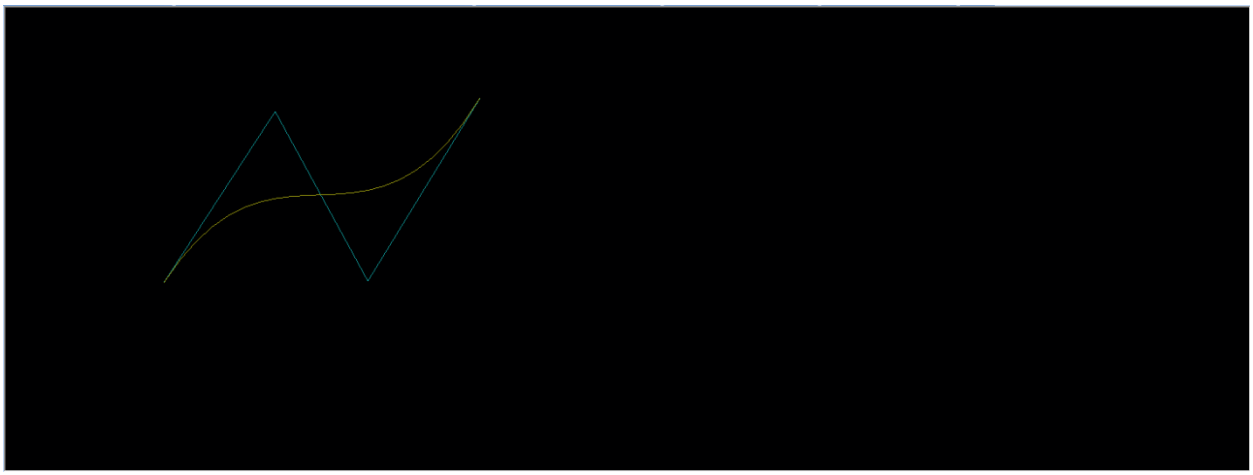
Control polygon and b-spline with more control points:



### Line ▸ Bezier

Draw Bezier by the set of points. To draw Bezier curve by control polygon four control points required.

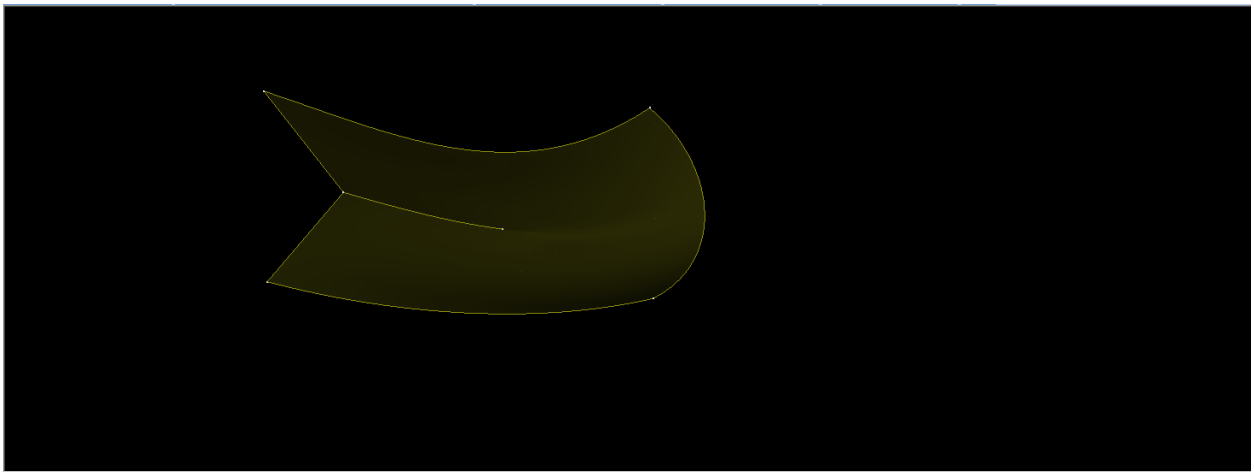
Input first point of the line by cursor or coordinates. Input tree more points. New Bezier will appear in the model. To stop input Beziers press Esc button or right mouse button. To stop Beziers input press Esc button or right mouse button one more time.



### Line ▸ Knuckle

Detect knuckle lines inside surface.

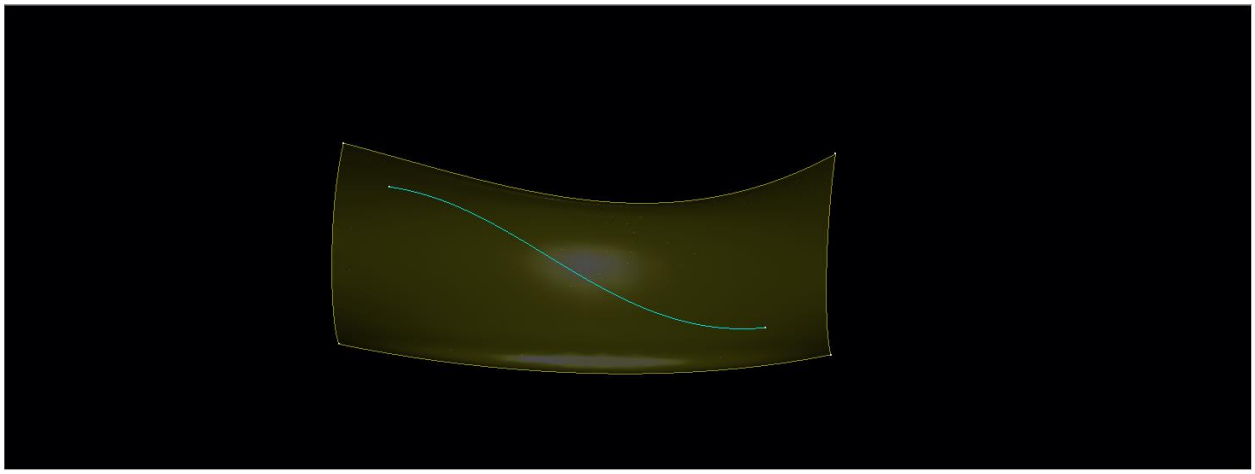
Select surface for detecting knuckle lines. Detected knuckle lines will appear in the model.



**Line ▸ Surface Line**

By this command user set up surface lines mode. All lines inputted in this mode lines will belong to current surface.

Line on surface:



## Draw ▸ Rectangle

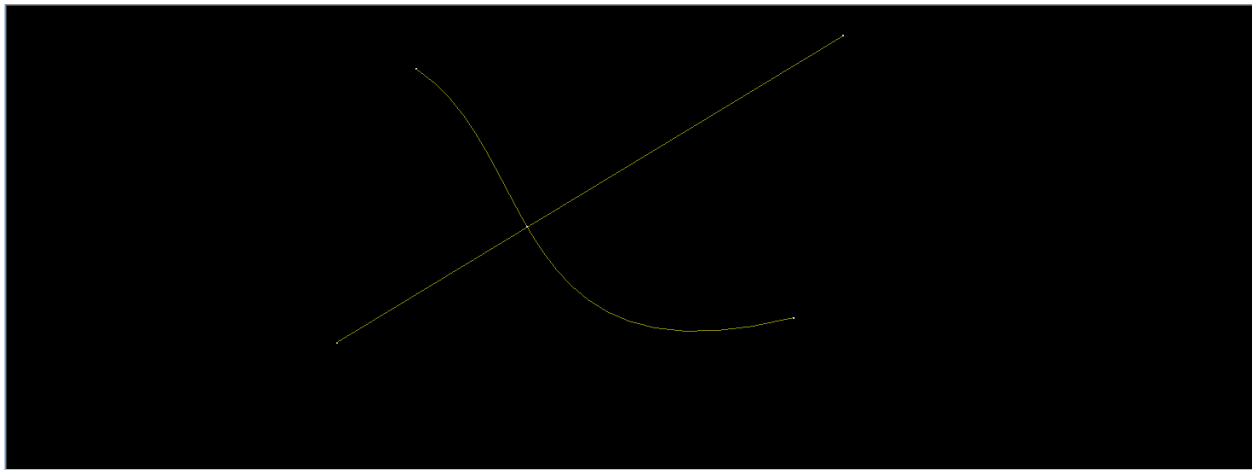
Draw rectangle between two diagonals points. The rectangle consist of four boundary lines and surface inside.

Input first point by cursor or coordinates. Input second point. Rectangle will appear on screen. Input new point to start new rectangle. To stop rectangles input press Esc button or right mouse button.

## Draw ▸ Point

Calculate intersection point between two lines.

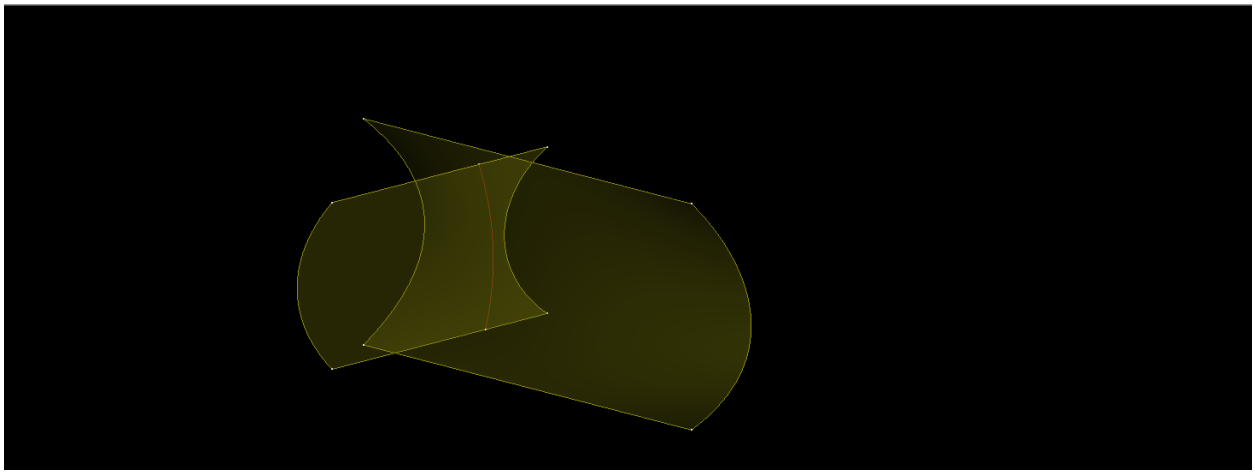
Select first line. Select second lines. Intersection point will appear on screen.



## Draw ▸ Intersection

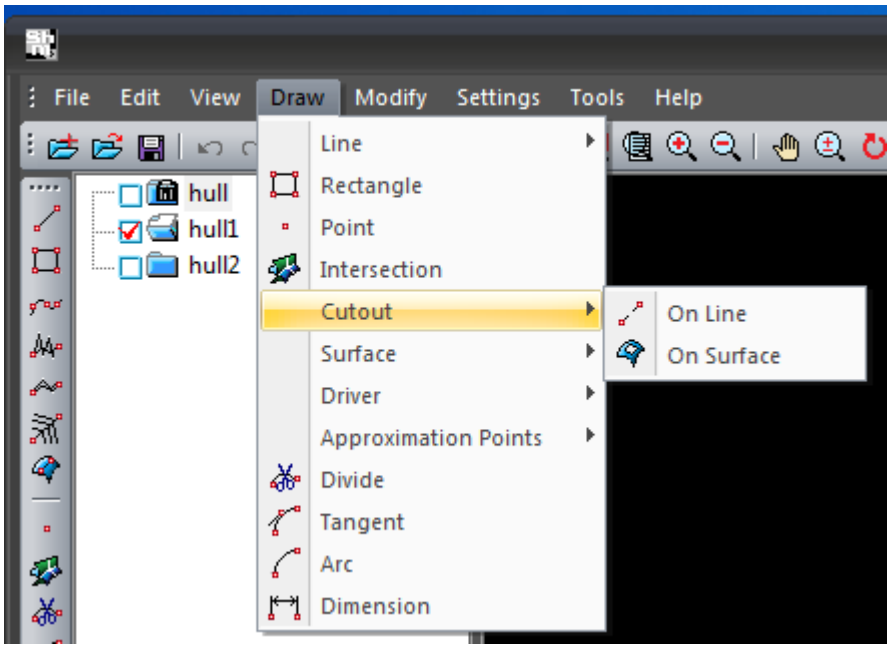
Calculate intersection line between two surfaces.

Select first surface. Select second surface. Intersection line will appear on screen.



## Draw ▸ Cutout

By this command user can make cutouts for lines and surfaces.

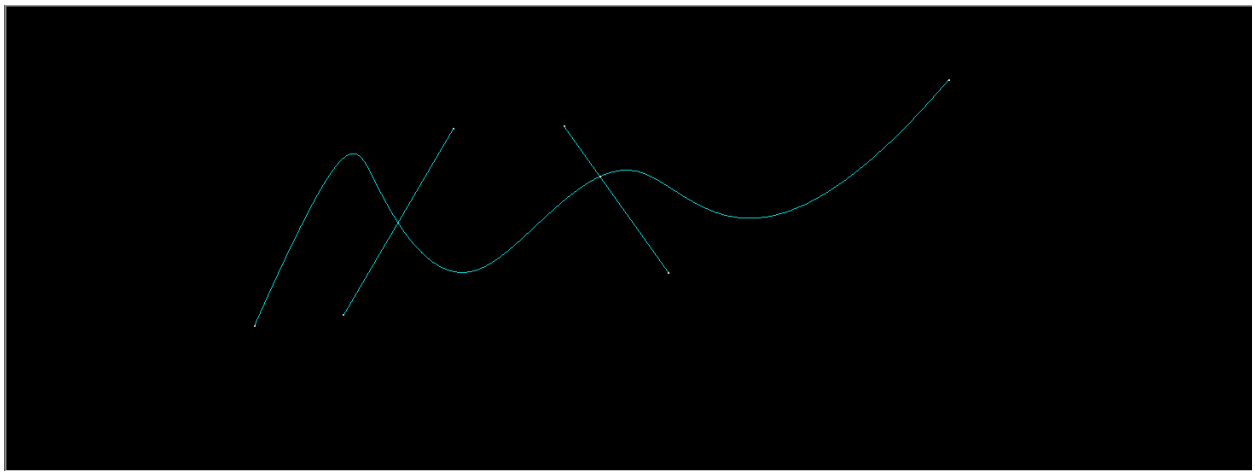


**Cutout ▸ On line**
Hide part or parts of the line.

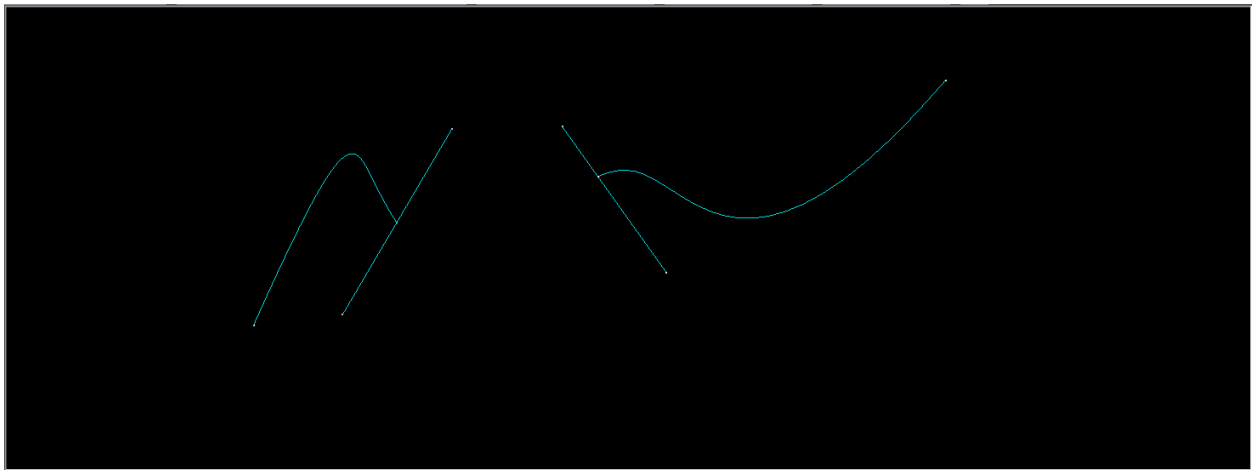
Select begin point on line to make cutout. Select end point. Part of the line between selected points will disappear from the screen.

Both selected points must belong to the same line.

Before:



After:



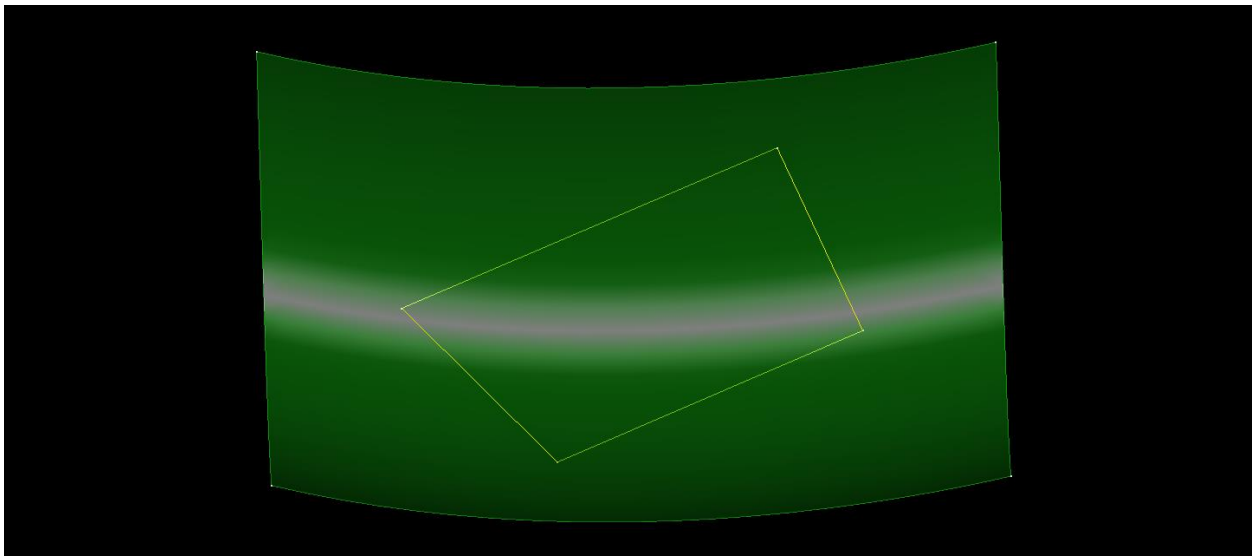
**Cutout ▸ On surface**
Hide part or parts of the selected surface.

Select line by line closed contour on the surface. Part of the surface inside cutout will disappear from the screen.

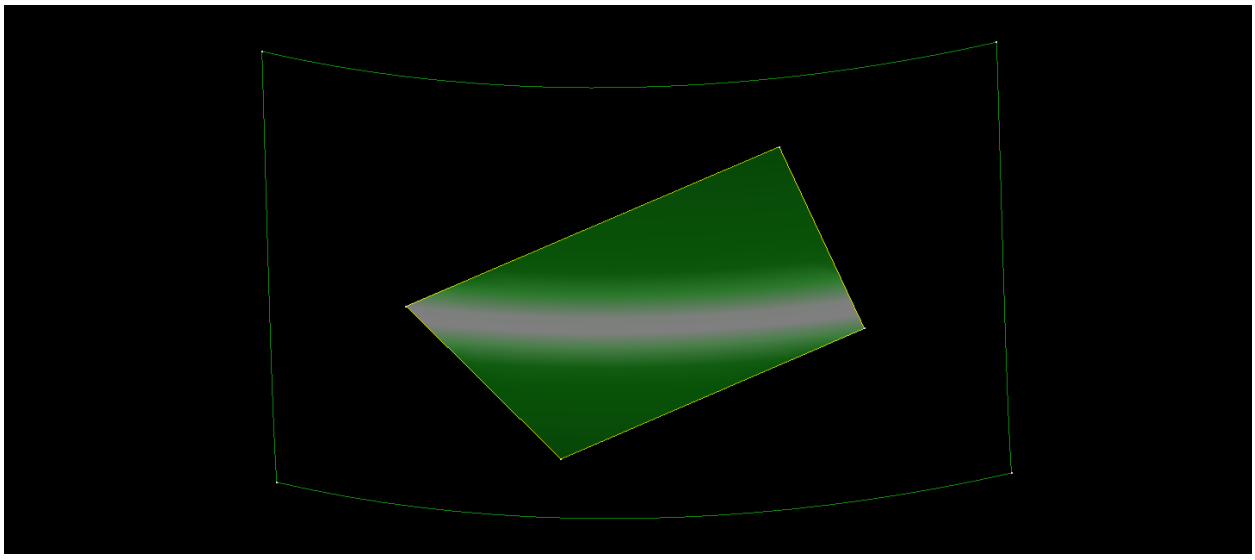
All selected lines must be connected each other and belongs to the same surface. Two cutouts cannot have same boundaries or intersections.

Current version of the Shape Maker support outer and inner contours of the trimming. Most outer contour is surface boundary contour. Inner contours holes. So if necessary to define hole – two contours should be presented: hole itself and outer contour as surface boundary.

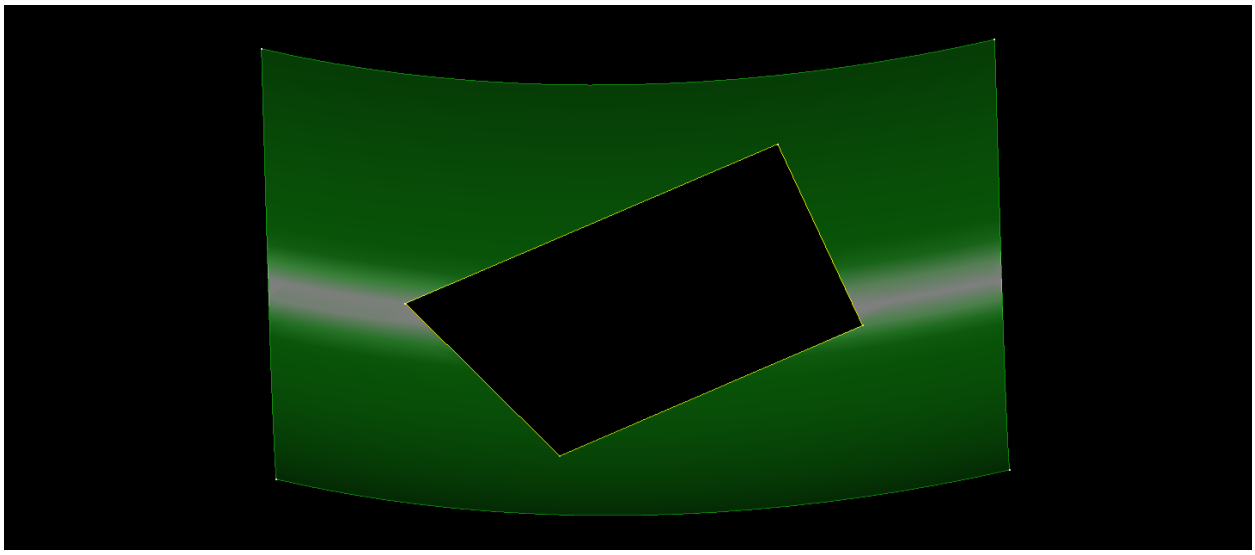
Before:



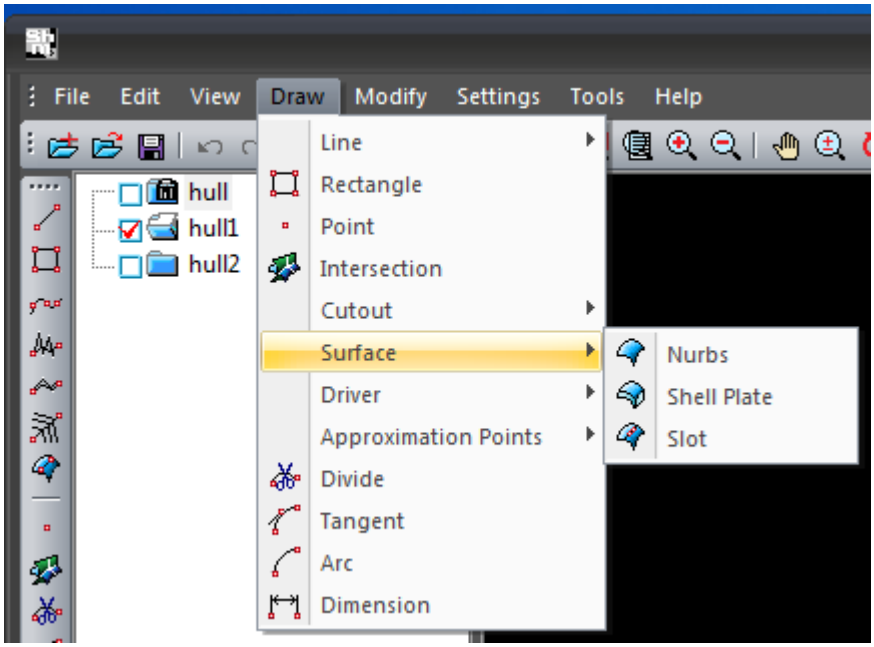
After making inner contour:



After making outer contour:



## Draw ▸ Surface



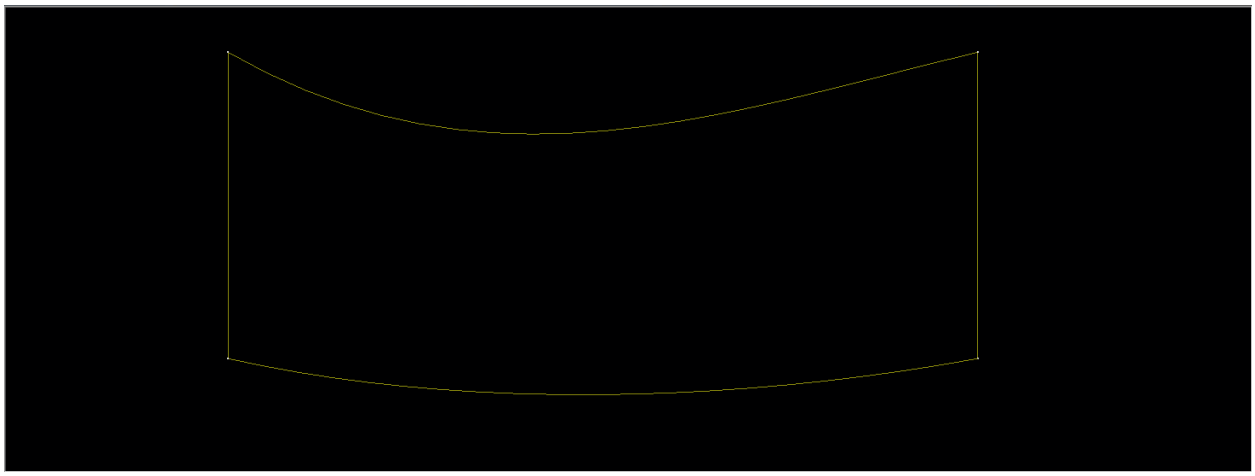

### Surface ▸ Nurbs

Draw Nurbs surface.

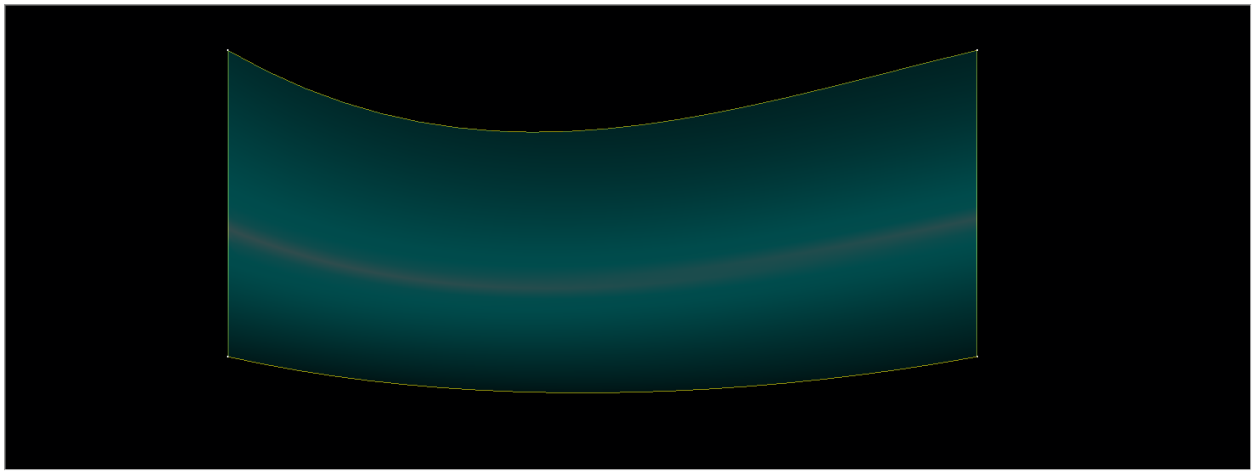
Select line by line closet contour. Nurbs surface will appear in the model.

Selected lines must be connected each other. Nurbs surface contour can contain from 2-4 boundaries.

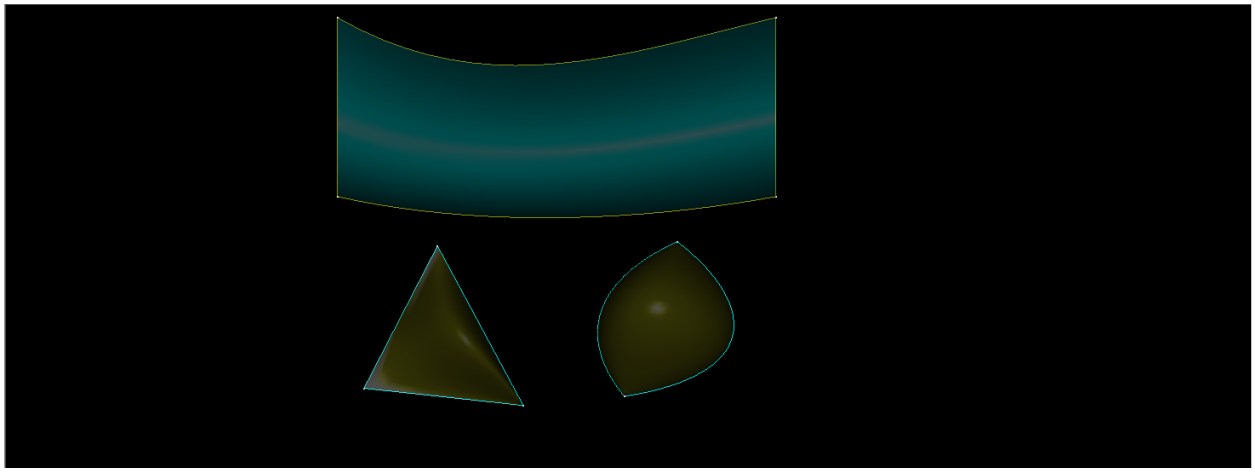
Before:



After:



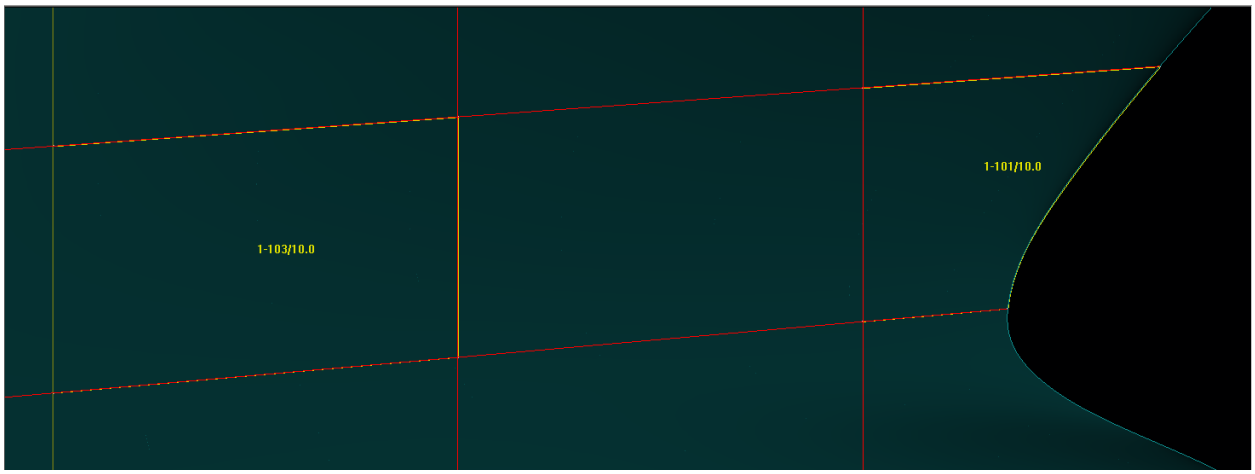
Possible number of the boundaries for Nurbs surfaces:



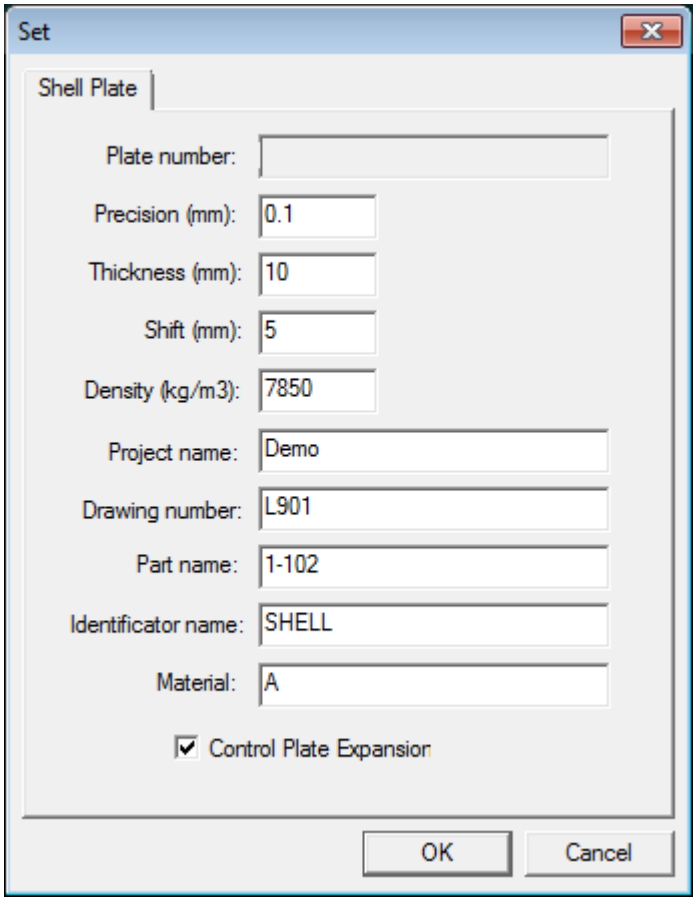
**Surface ▸ Shell Plate**

Define ship hull shell plates on the set of hull surfaces.

Select line by line contour of the shell plate and press Enter.




Fill all necessary field in Shell plate definition menu:




Precision - development precision.

Thickness - plate thickness.

Shift - plate thickness deflection.

Density **-** material density.

Project name - parameter for Foran system.

Drawing number - parameter for Foran system.

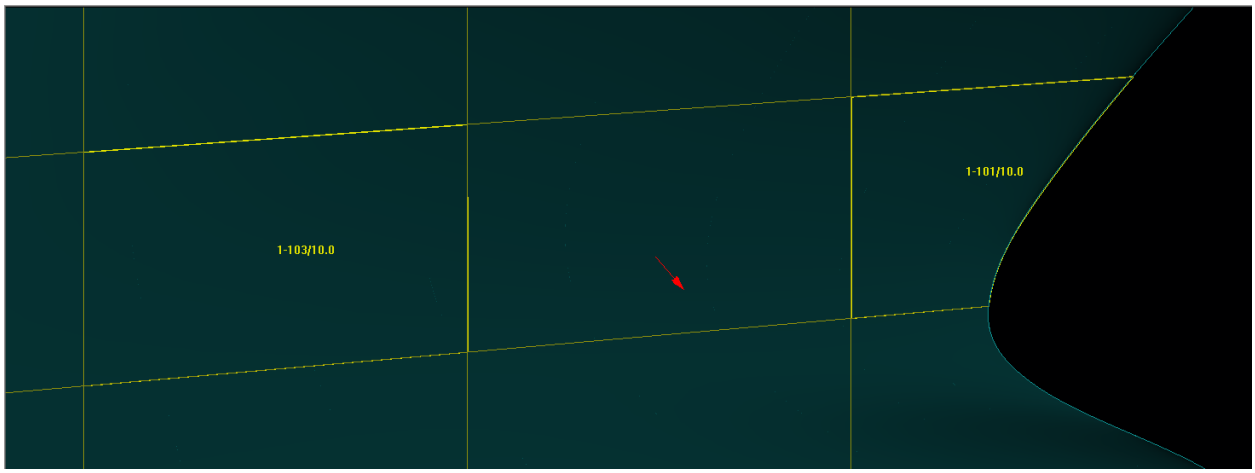
Part name (Foran panel) - part number.

Identificator name - comments for the name of the part.

Material **-** material grade.

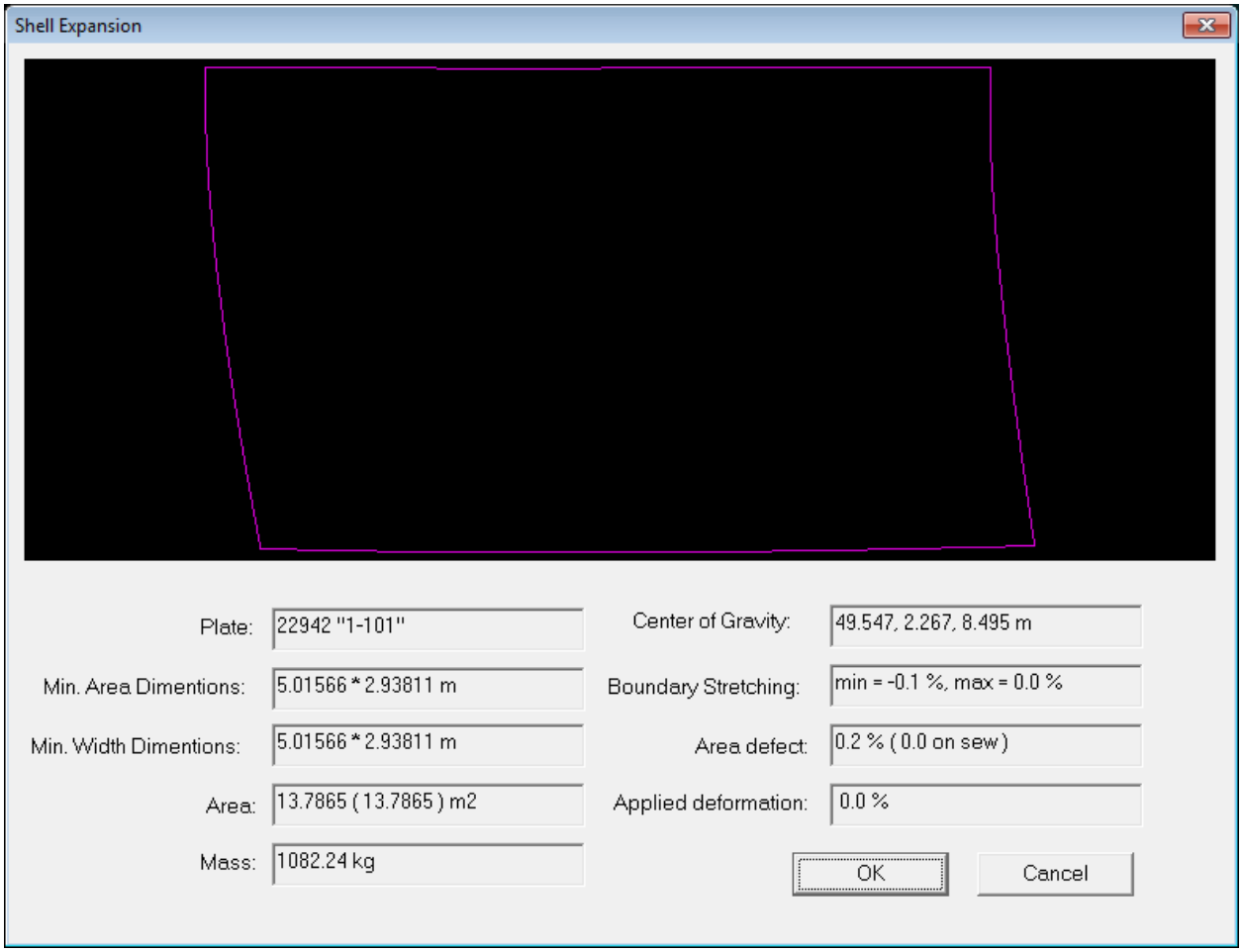
Control Expansion – visual checking shell plate expansion.

Select orientation and press Enter.




Click on plate if necessary to change proposed thickness orientation. Then press Enter.

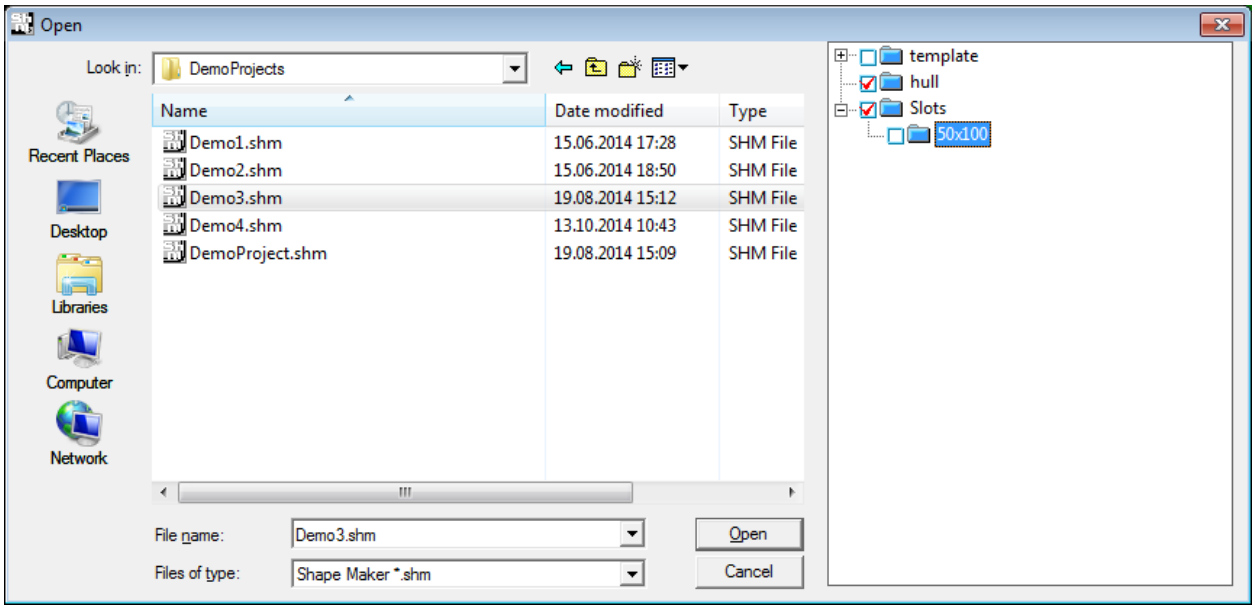
If Option Control Expansion was switch on following dialog will appear:



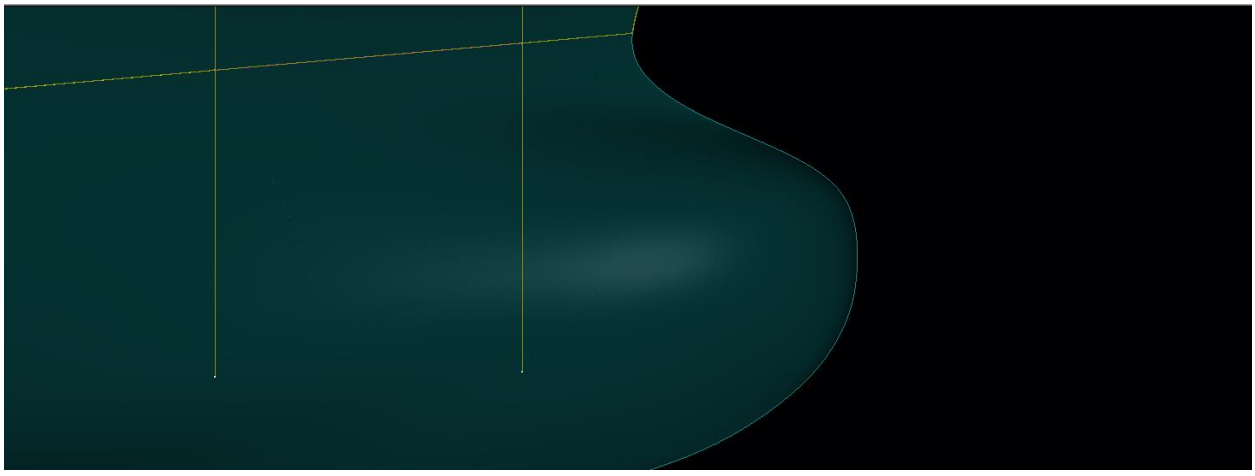

### Surface ▸ Slot

Define set of similar cutouts on surface along surface line. Used for slot welding marking. Geometry of the cutouts should be predefined in separate block.

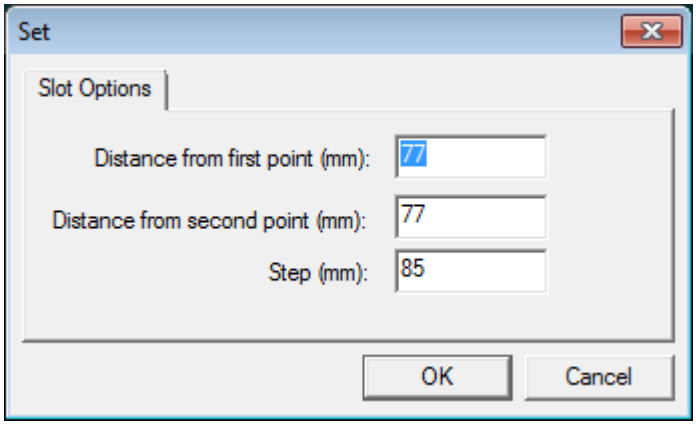
Open slots library, choose required slot and then click Open.



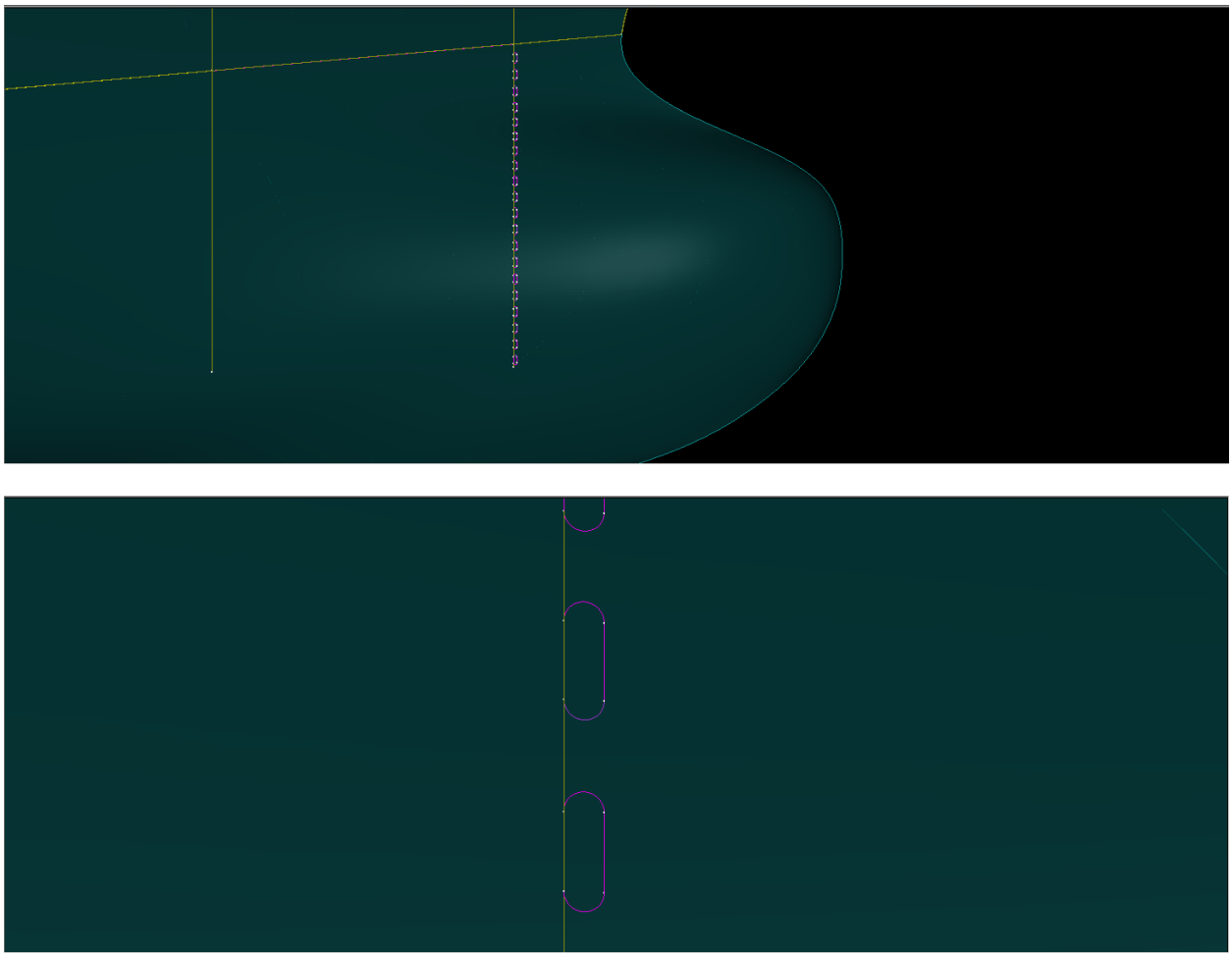

Select begin and end points on the surface line.



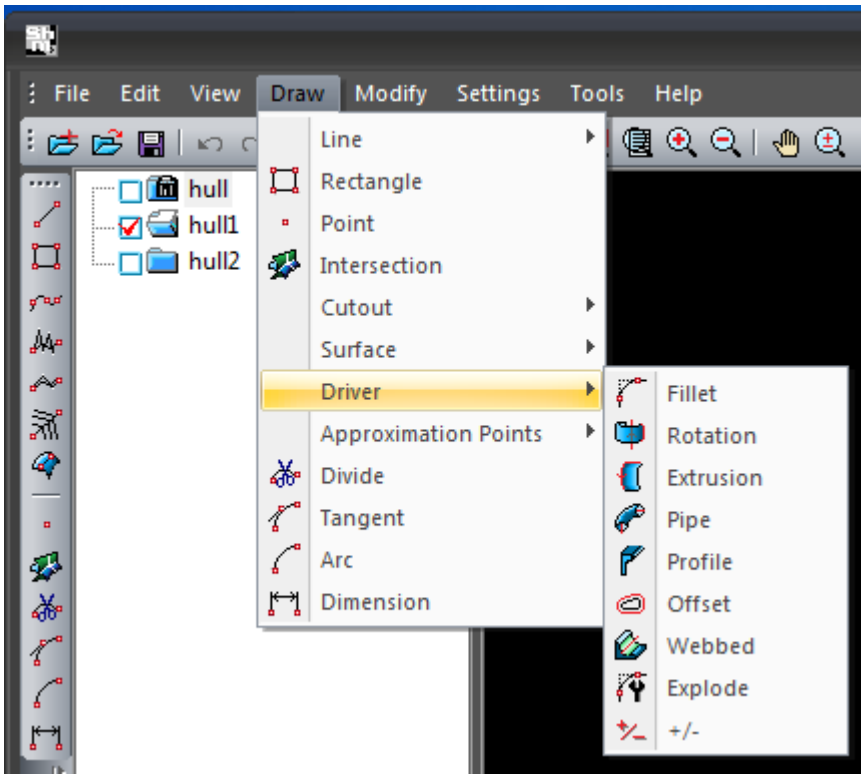
Set necessary parameters and click OK:



Slots will appear in the model.



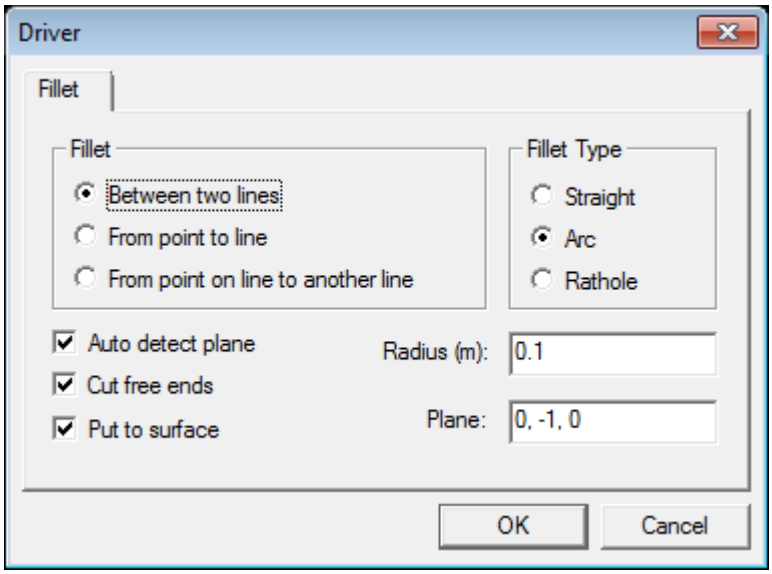
## Draw ▸ Driver



Drivers are a set of element build in according predefined rules: Surface of extruding, rotation and etc. User can edit drivers by changing parameters or geometry of elements which driver depend.

**Driver ▸ Fillet**

Fillet is used for obtain a smooth conjugation of two lines by a line of an arc or a straight line.




Select the fillet construction method and other required options.

There are three methods for creating a fillet.

Between two lines

From point to line

From point on line to another line

Fillet type**:**

Straight – straight line fillet;

Arc – arc fillet;

Rathole – rathole fillet;

Auto detect plane - automatic detection of the projection where the fillet will be of an arc shape:

Cut free ends - cutting the “free” ends of the mated lines.

Put to surface – make a surface line fillet.

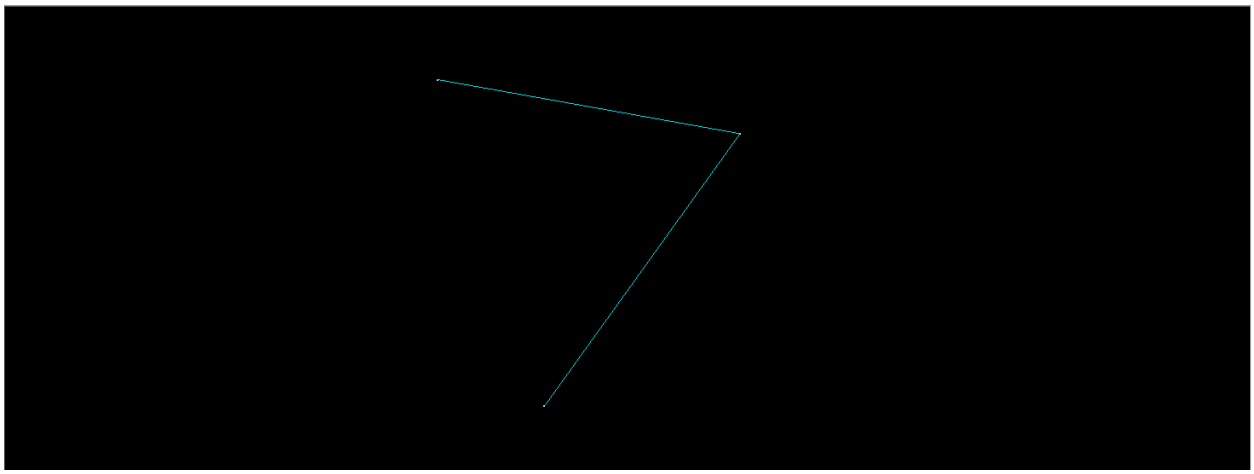
Radius – value the arc radius.

Plane – normal to projection plane, on which the line should be of an arc shape.

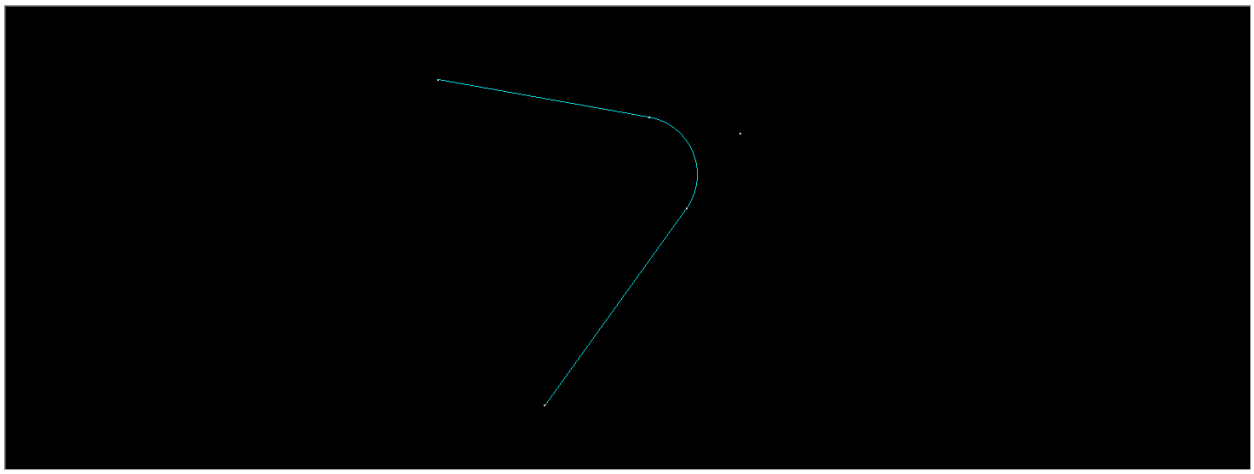
Fillet between two lines:

Select first line. Select second line. Fillet driver will appear in the model. To stop creating the fillet press Esc or right-click the mouse

Before:



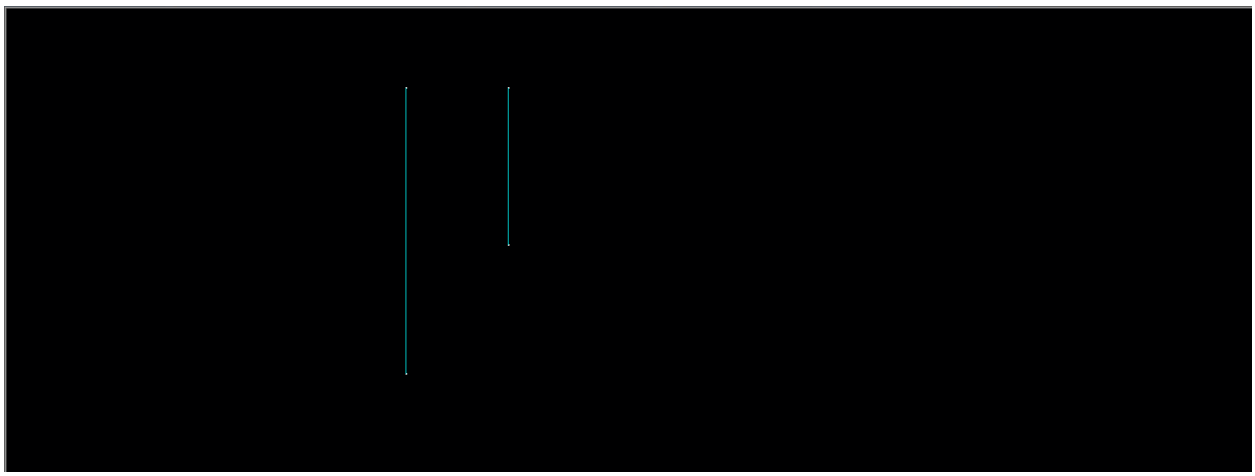
After:



Fillet from point to line:

Select the starting point of the fillet line. Select line to which the fillet will be created. To stop creating the fillet press Esc or right-click the mouse.

Before:



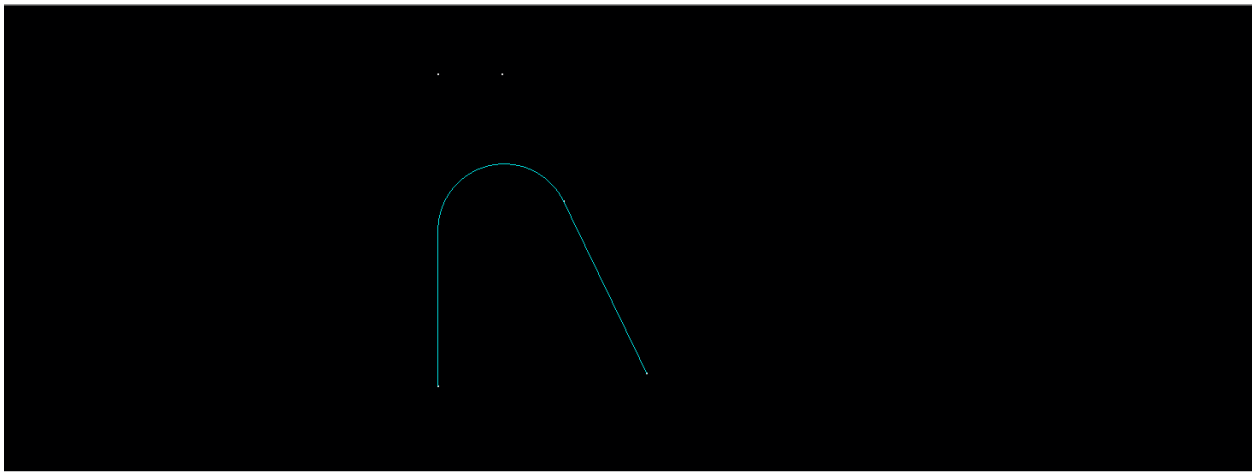
After:

Fillet from point on line to another line. Radius will be calculated automaticly.

Select the starting point of the fillet line. Select the line, to which the fillet will be created. To stop creating the fillet press Esc or right-click the mouse.

Before:

After:

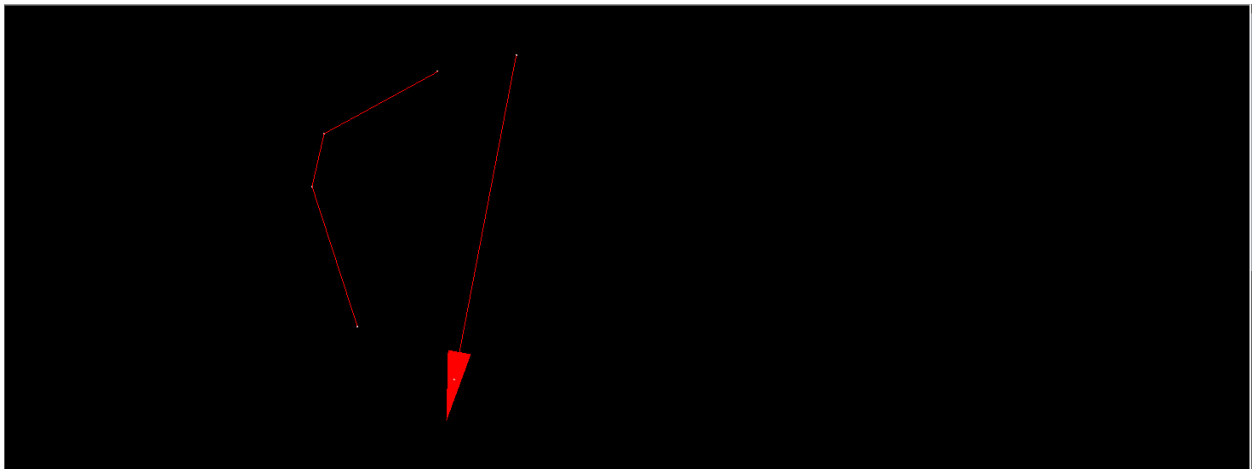


### Driver ▸ Rotation

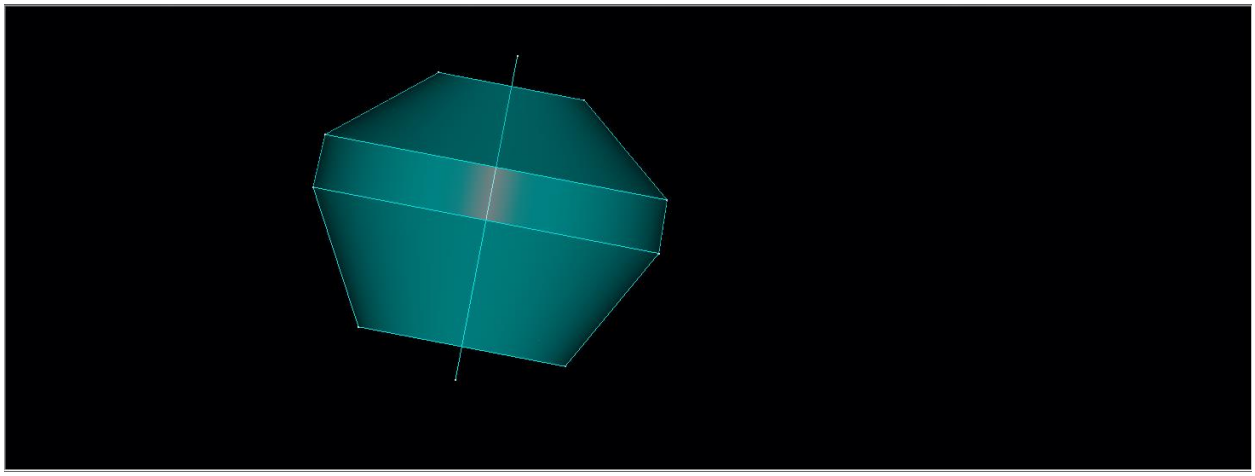
Rotation is used to create surfaces by rotation of the forming lines around the axis.

Select the axis of rotation. Select a set of lines for rotation. Enter the angle of rotation. To stop creating the driver of rotation press Esc or right-click the mouse.

Before:



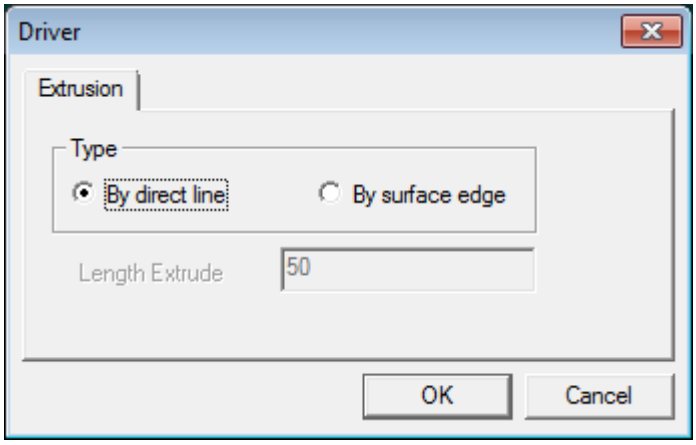
After:



### Driver ▸ Extrusion

Extrusion is used to create surfaces by extrusion the forming lines along the directional line.

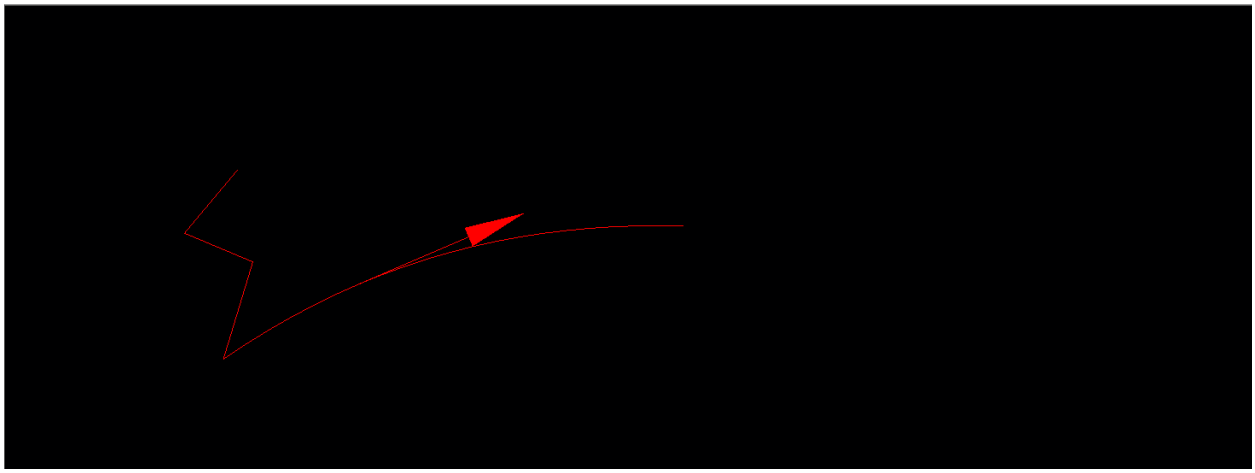
Select the method of extruding.



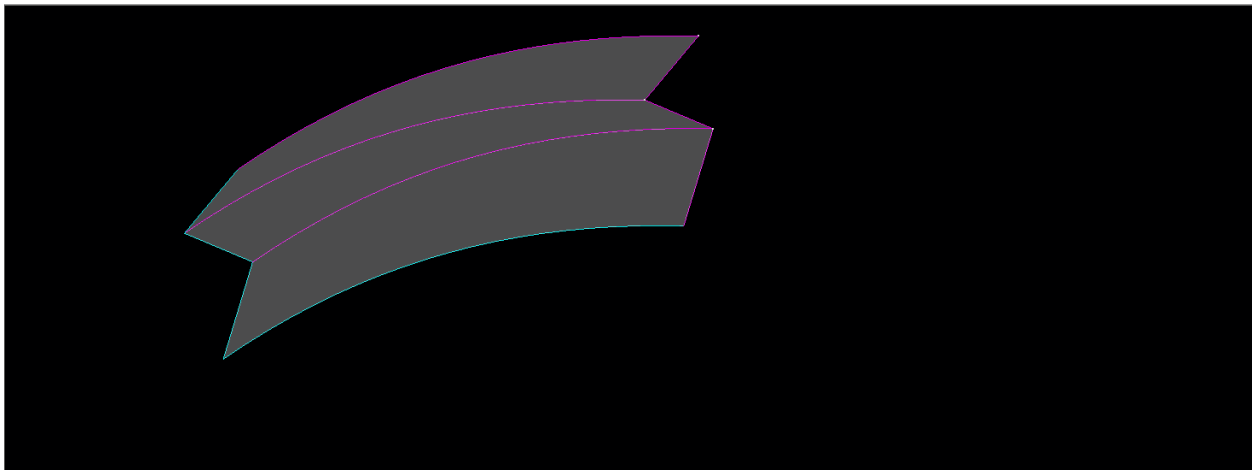

By direct line:

Select the directional line. Select a set of lines for extrusion. To stop creating the driver of extrusion press Esc or right-click the mouse.

Before:



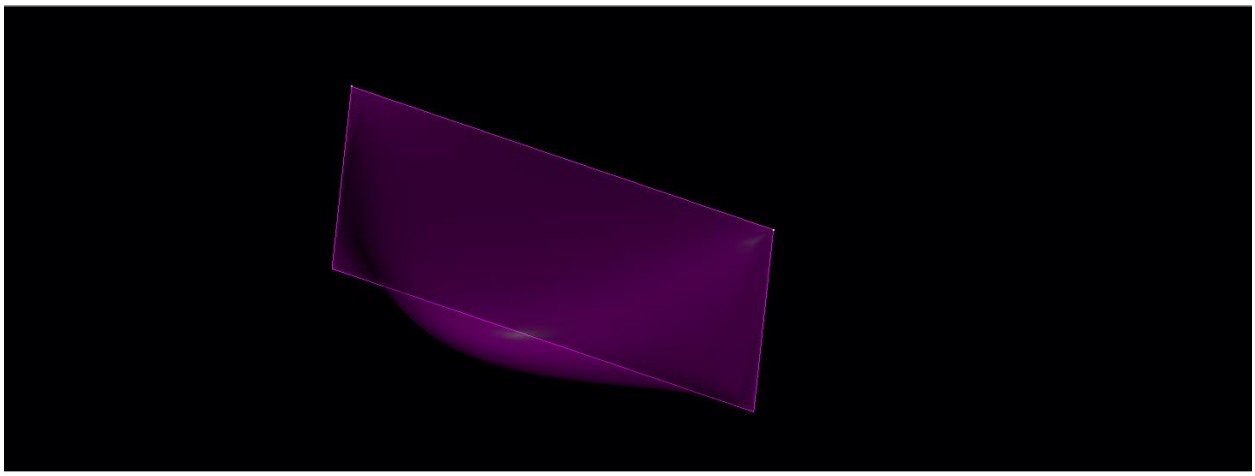
After:



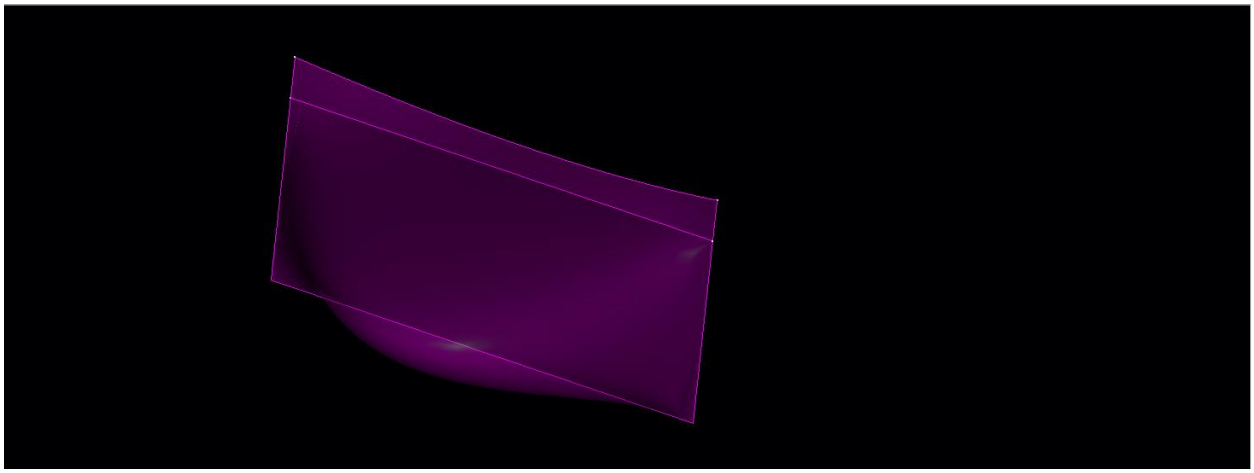
By surface edge:

Select boundary of the surface. Extension of the surface is created to the length specified in the dialogue box.

Before:



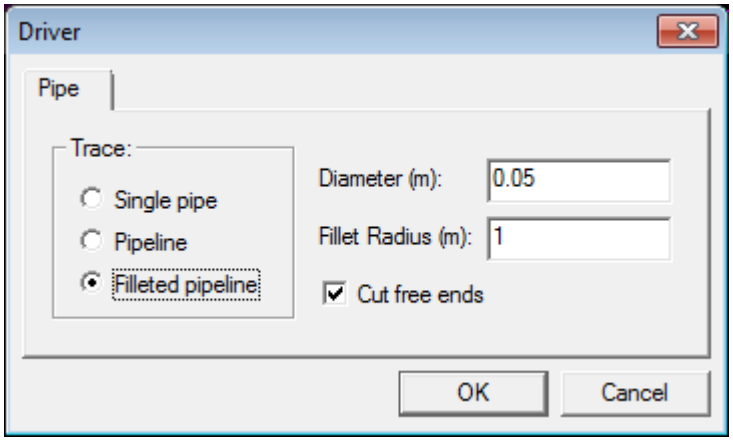
After:



**Driver ▸ Pipe**

Pipe is used for development of pipelines.

Select the pipe construction method, as well as desired parameters and options.




Trace: method for tracing the pipe:

Single pipe – created on a segment of a single line;

Pipeline – created on a chain of lines;

Fillet pipeline – created on a chain of lines;

Diameter – diameter of the pipe;

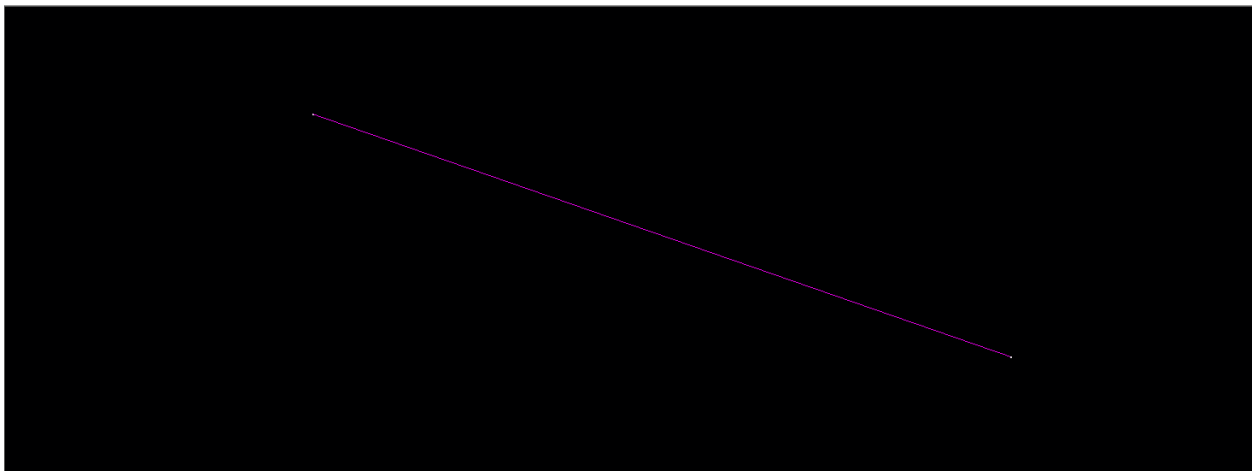
Fillet Radius – fillet radius of the trace line;

Cut free ends - cut free ends of the lines;

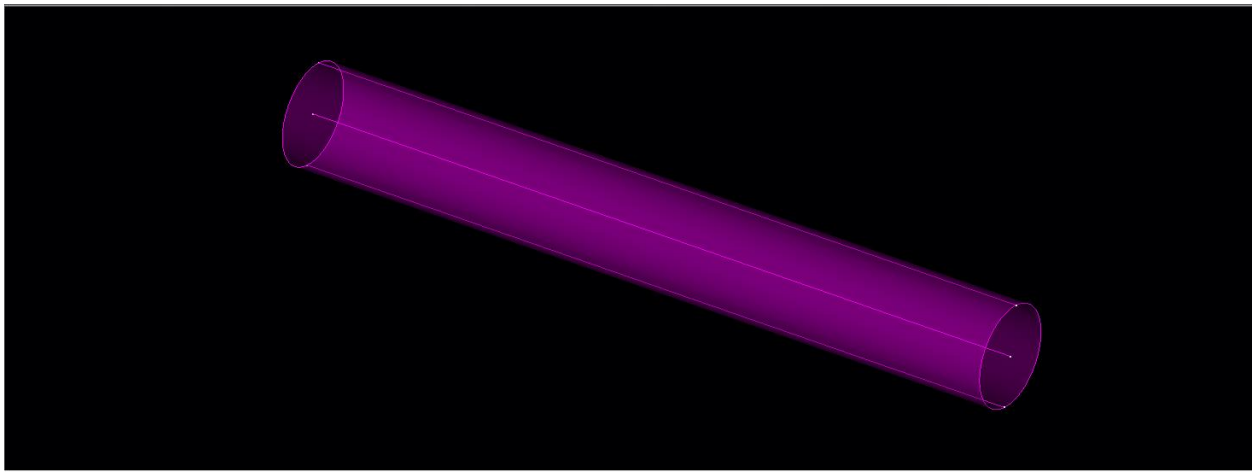
Single pipe:

Select a first and end points of the line segment. A pipe surface is created. To stop creating the pipe press Esc or right-click the mouse.

Before:



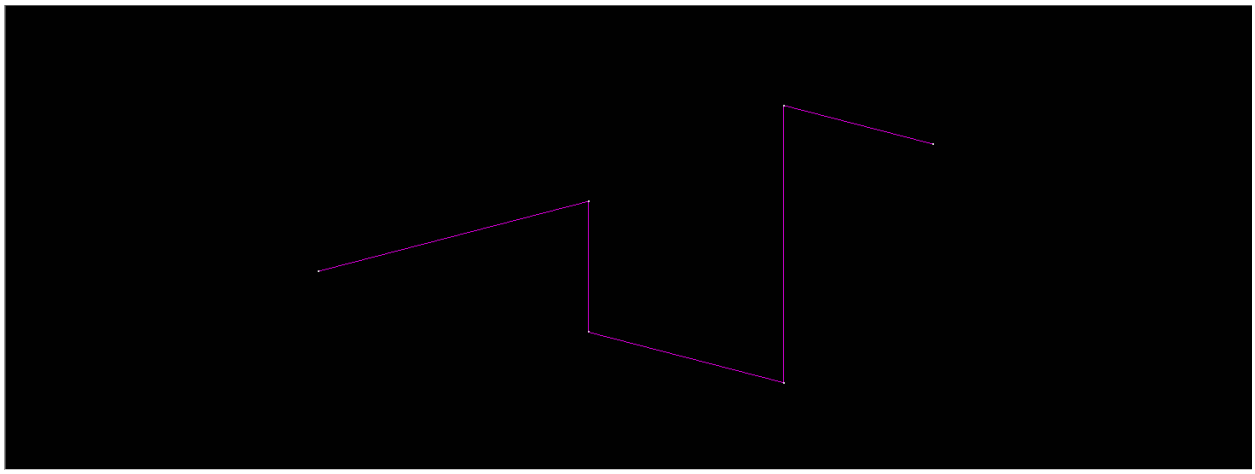
After:



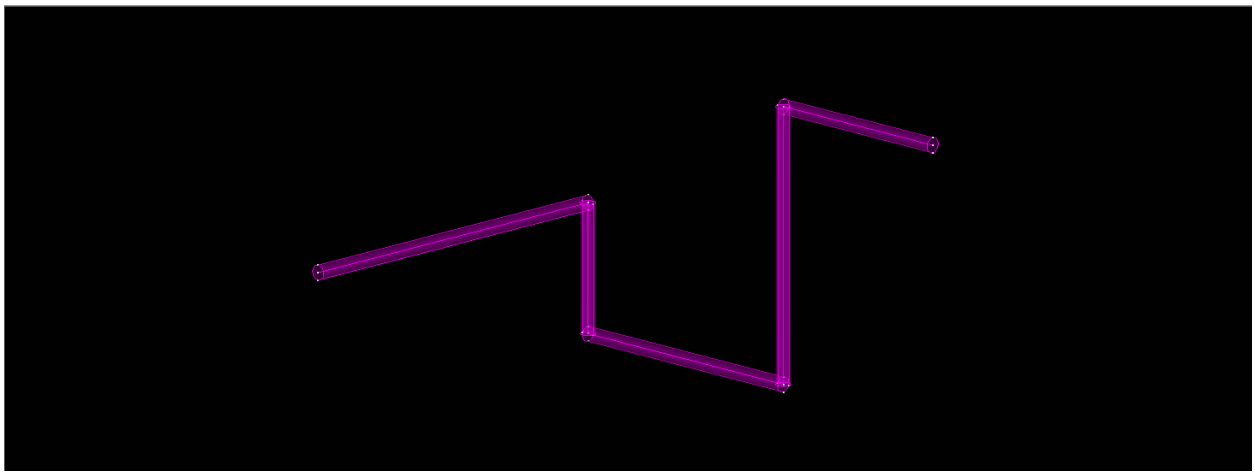
Pipeline:

Select begin point. Select end point. Pipeline will appear in the model. To stop creating the pipeline press Esc or right-click the mouse.

Before:



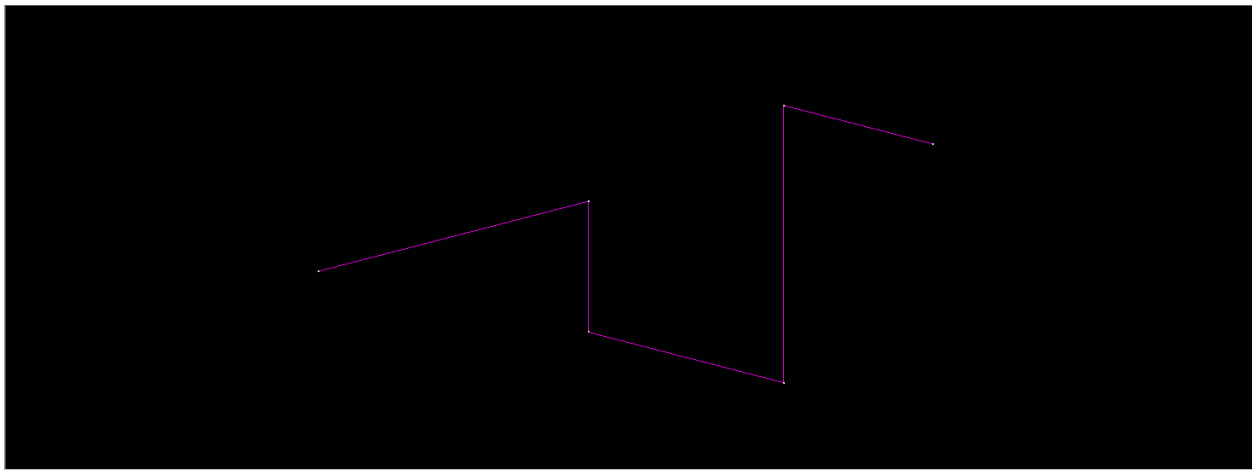
After:



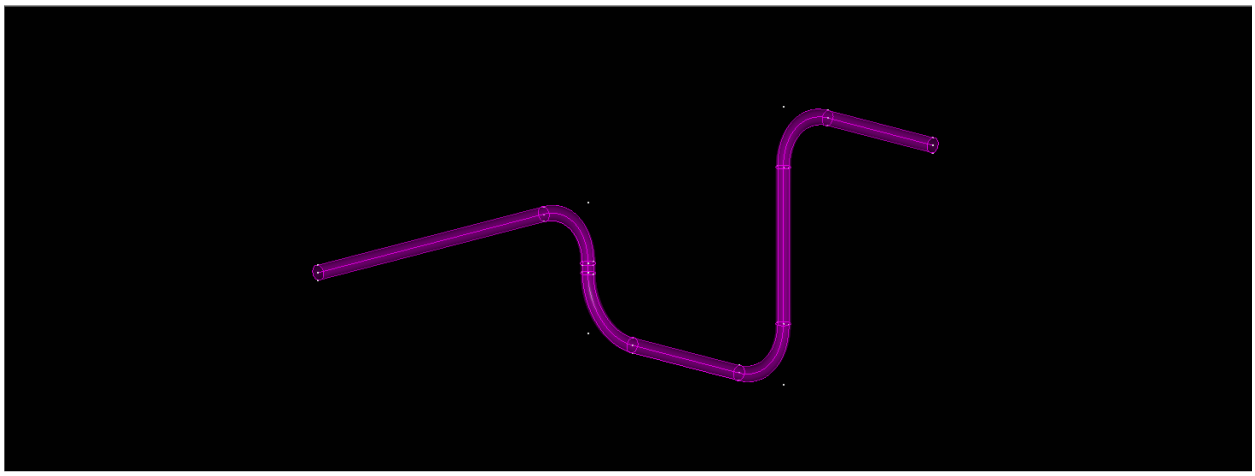
Fileted pipeline:

Select begin point. Select end point. Pipeline will appear in the model. To stop creating the pipeline press Esc or right-click the mouse.

Before:



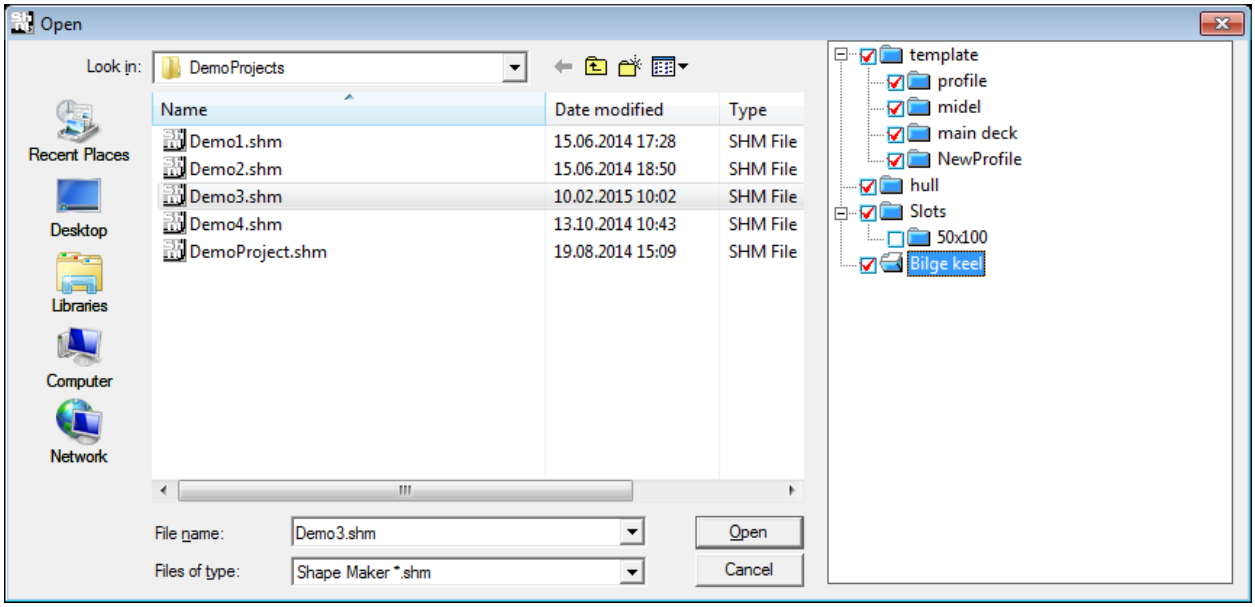
After:



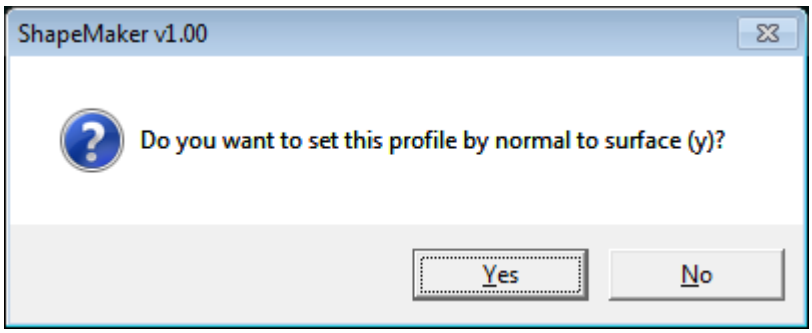
**Driver ▸ Profile**

Profile is used for creating profile surfaces by tracing the profile along the directional line. Geometry of the profile should be predefined in separate block.

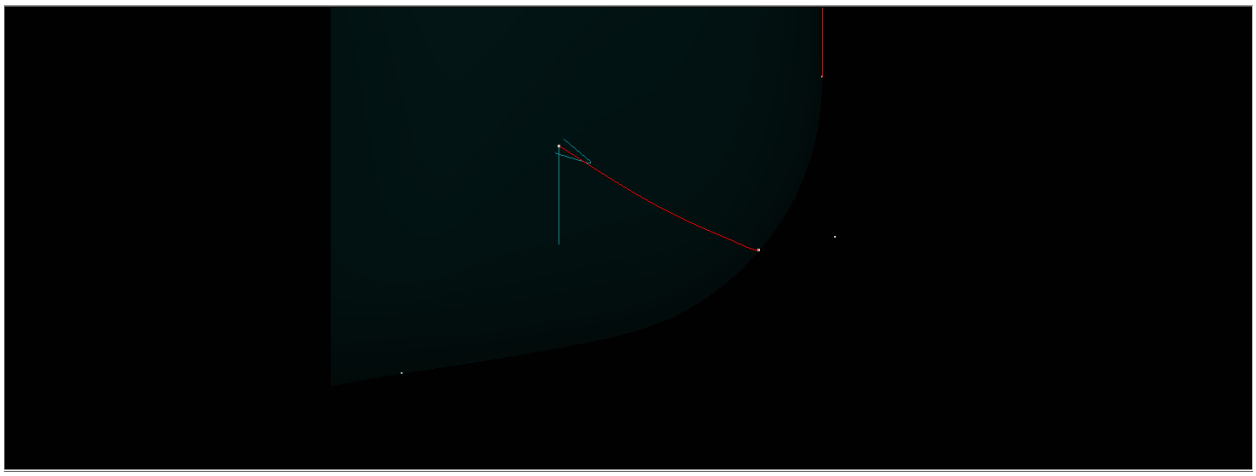
Select block with profile geometry:



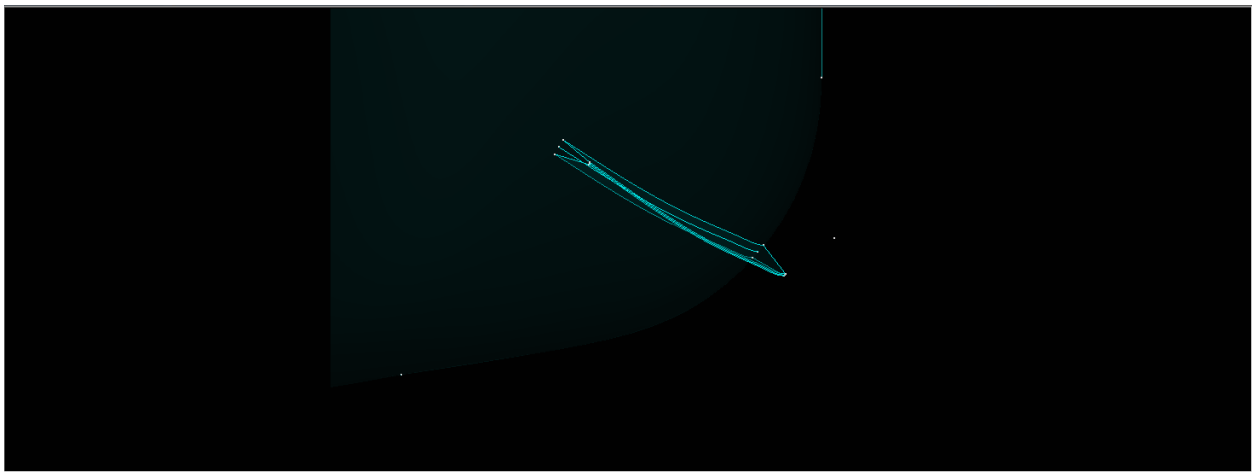

Then click Open. Select begin and end points of the directional line. Answer Yes if profile shoul be normal to surface.




Select profile orientation:



Profile will appear in the model.



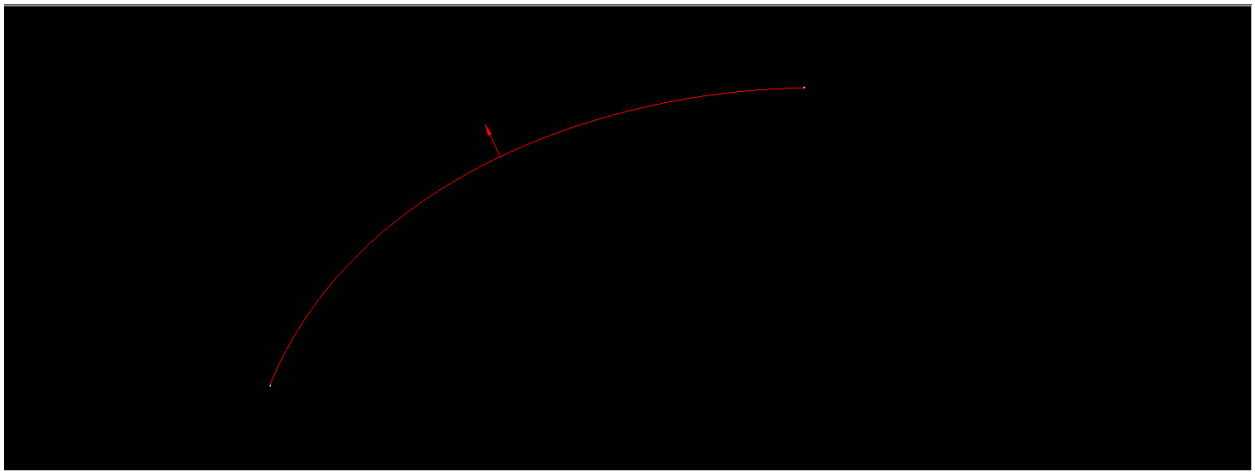
To stop creating the profile press Esc or right-click the mouse.

### Driver ▸ Offset

Offset is used to create a line or a surface equidistant from another line or surface.

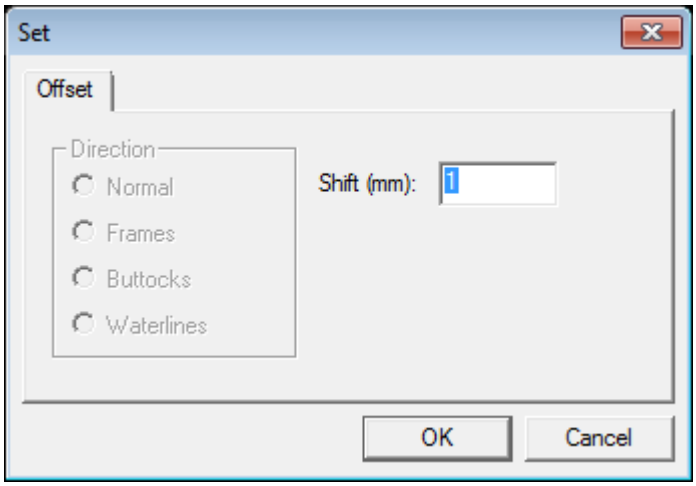
Line:

Select line for offset.



Click on selected line one more time to change offset direction. Press Enter.

Fill offset parameters in dialogue box:




Direction:

Normal – offset direction by normal to original surface.

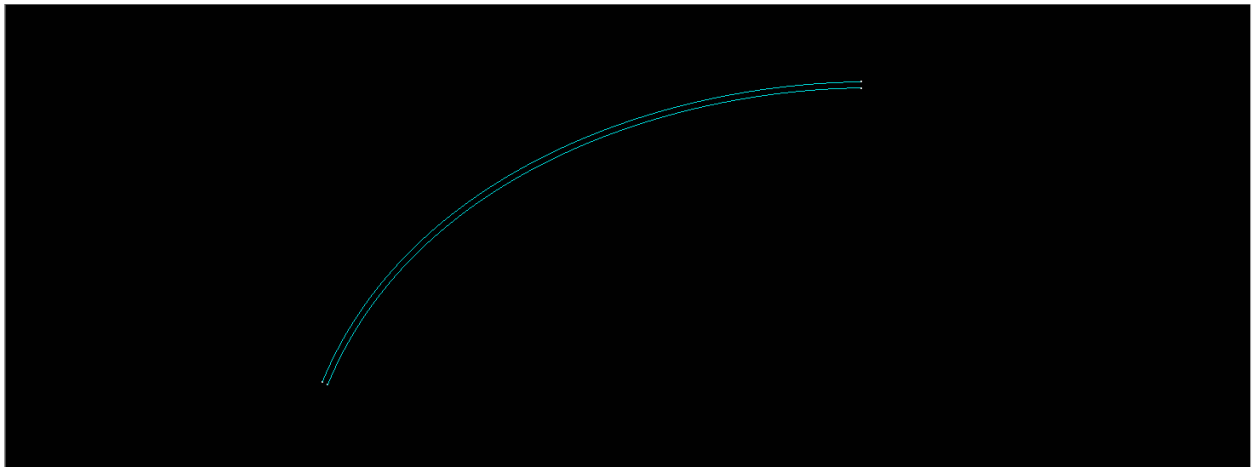
Frames – offset direction by normal to original surface on front projection.

Buttocks – offset direction by normal to original surface on side projection.

Waterlines – offset direction by normal to original surface on plan projection.

Shift – offset value.

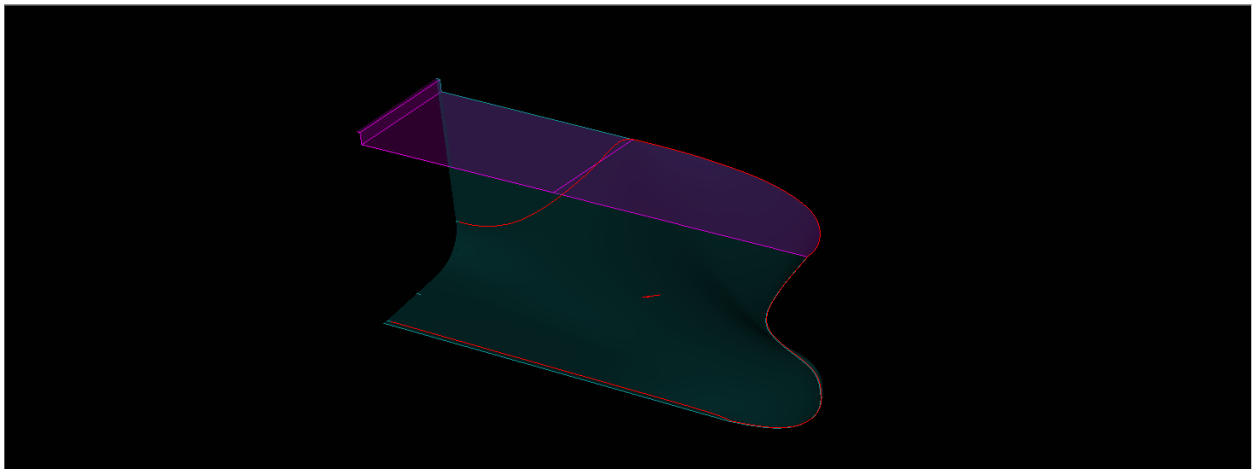
Offset line will appear in the model.



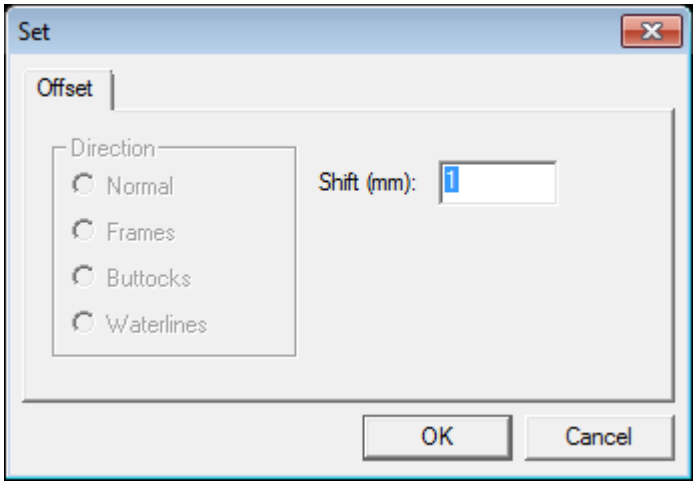
To stop creating the offset press Esc or right-click the mouse.

Surface:

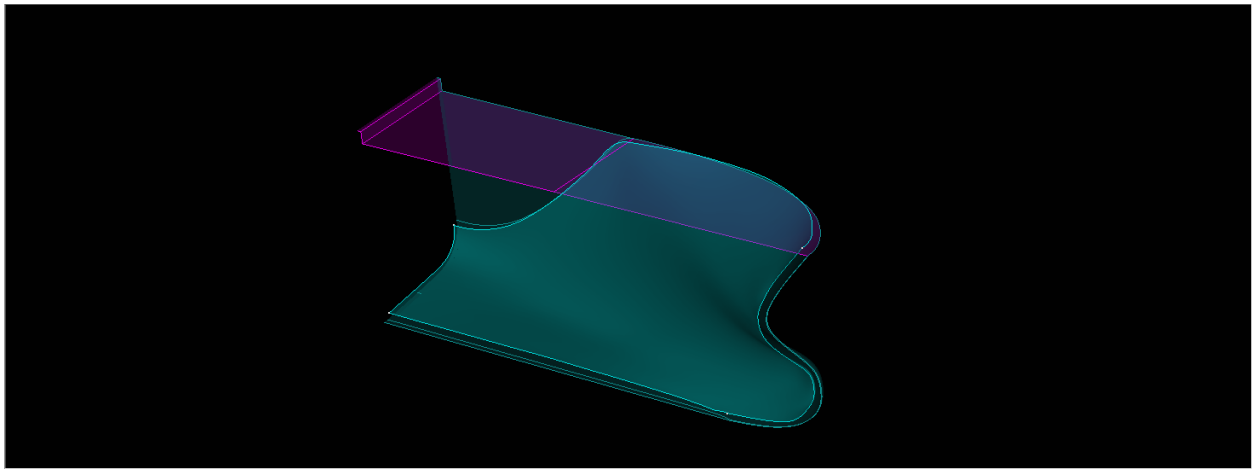
Select surface for offset.



Click on selected line one more time to change offset direction. Press Enter. Fill offset parameters in dialogue box:




Offset surface will appear in the model.



To stop creating the offset press Esc or right-click the mouse.

## Driver ▸ Webbed

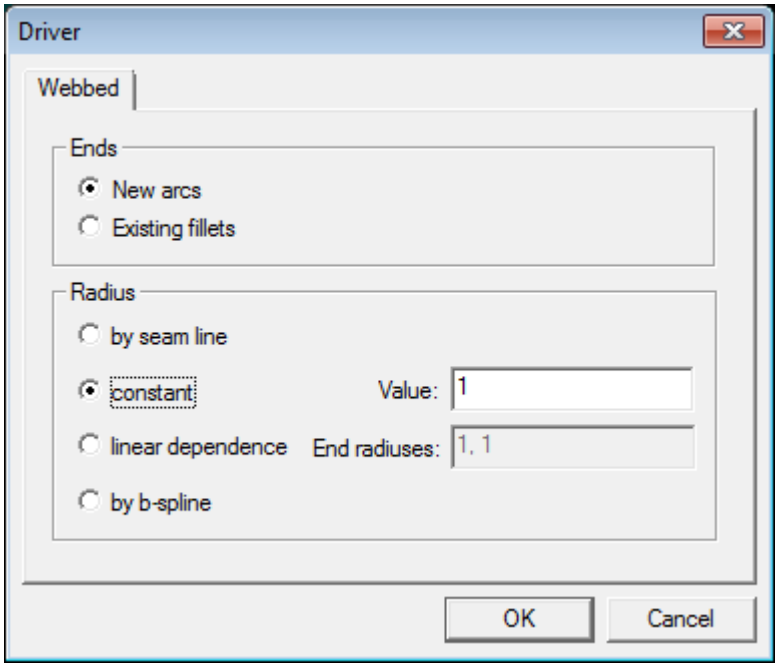
Webbed is used to create a smooth rounding edge of two surfaces.

In the dialogue box select the construction method and other required options to create a conjugation surface.

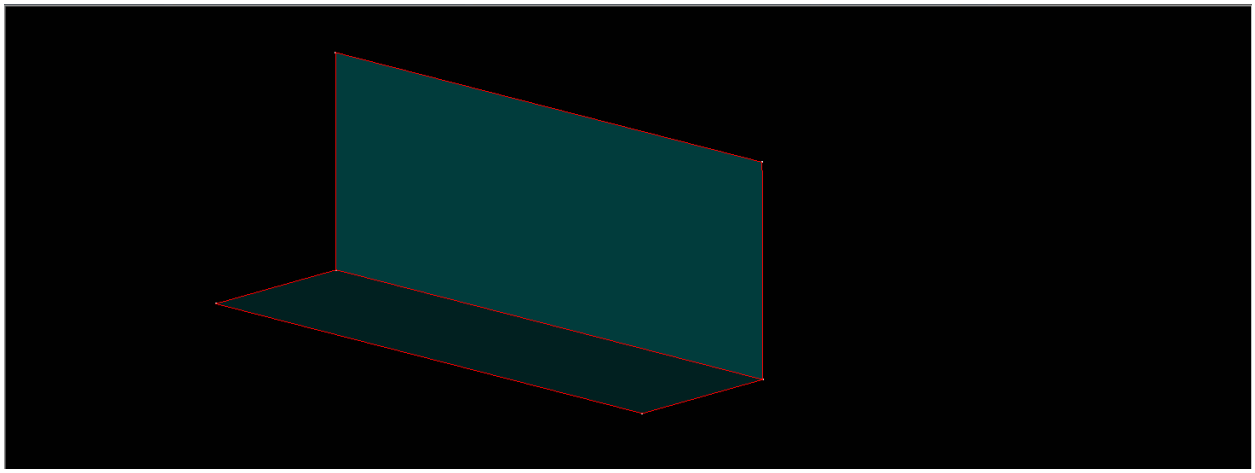
To create a conjugation it is necessary to select the surfaces to be rounded and the axis line. The rounding surface is created in the way that its section by any surface, intersecting the axis line parallel with it, is of an arc shape. In the end points such arcs are smoothly jointed with the mated surfaces. Arcs that correspond to beginning and end of the axis line are the butt edges of the mating surface.

For method constant (New arcs):

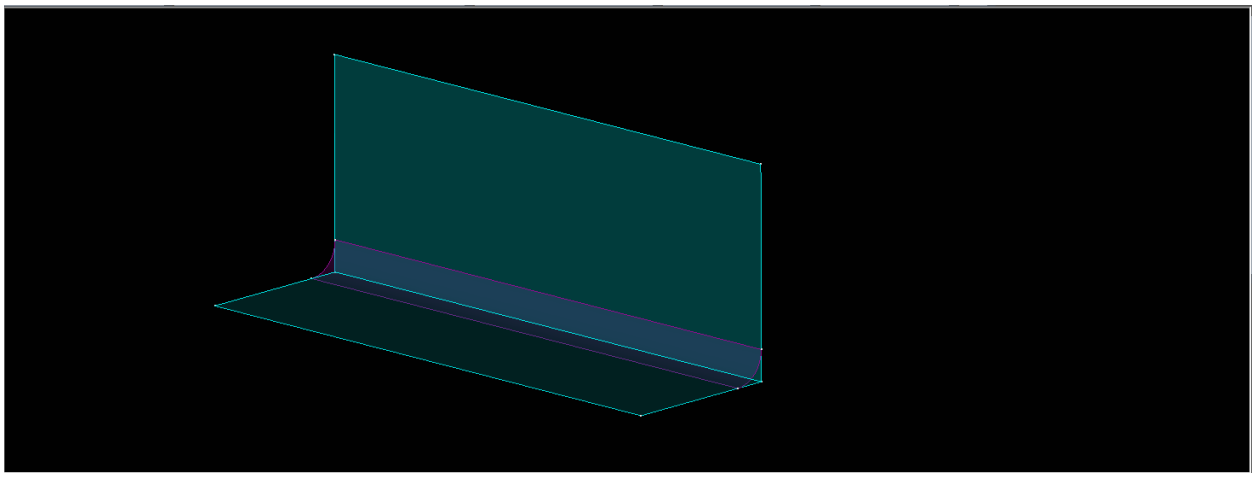
In the text field Value enter the conjugation radius. Confirm the selected value by clicking OK.




Select the first surface. Select the second surface. Select track line.



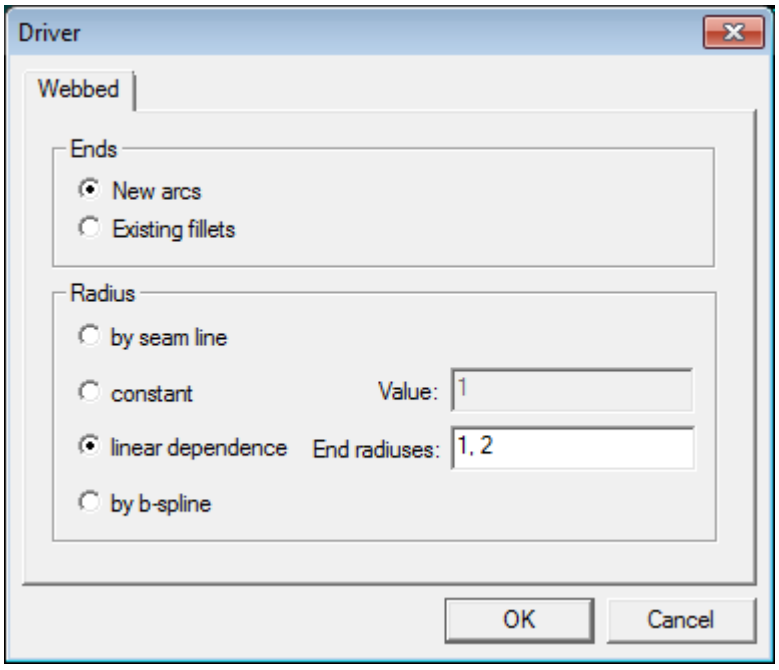
Webbed will appear in the model.



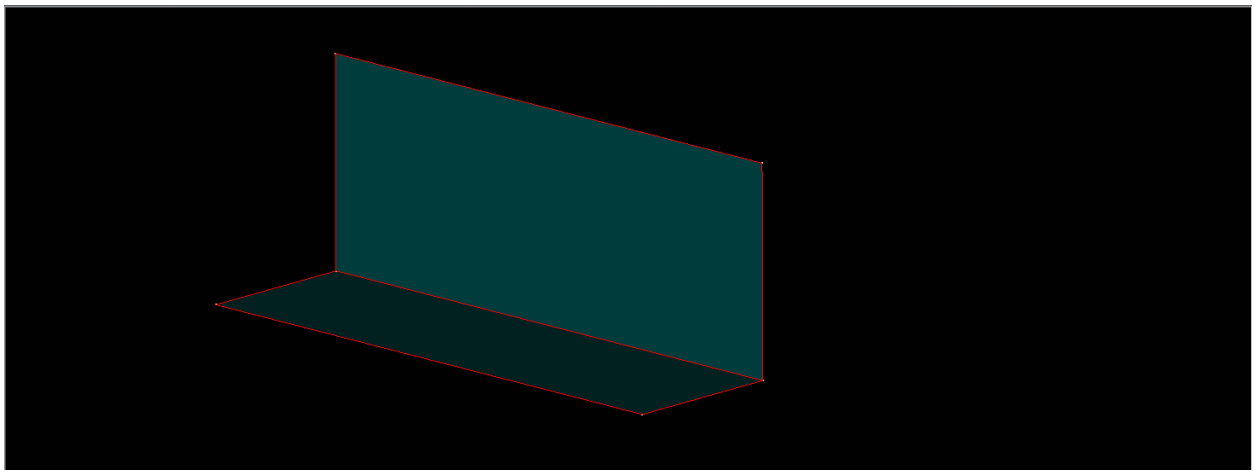
To stop creating the webbed press Esc or right-click the mouse.

For method linear dependence (New arcs):

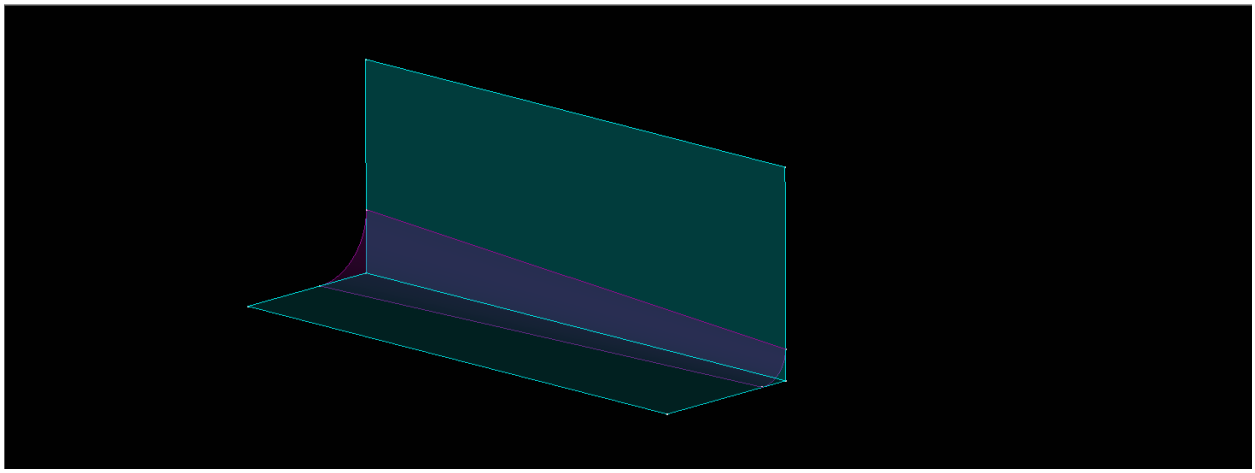
In the text field Radiuses enter the starting and ending radius of conjugation. Confirm the selected value by clicking OK.



Select the first surface. Select the second surface. Select track line.



Offset surface will appear in the model.



To stop creating the webbed press Esc or right-click the mouse.

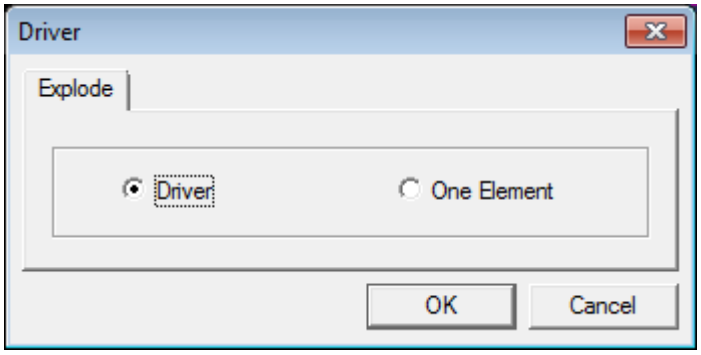
**Driver ▸ Explode**

This command is used to “explode” the driver or some elements of the driver.

After exploding all elements of the driver become independent elements.

Exploding the element will make it driver-independent: the element can be corrected as a usual element but cannot be deleted.

In the dialogue box select the driver “stripping” method.



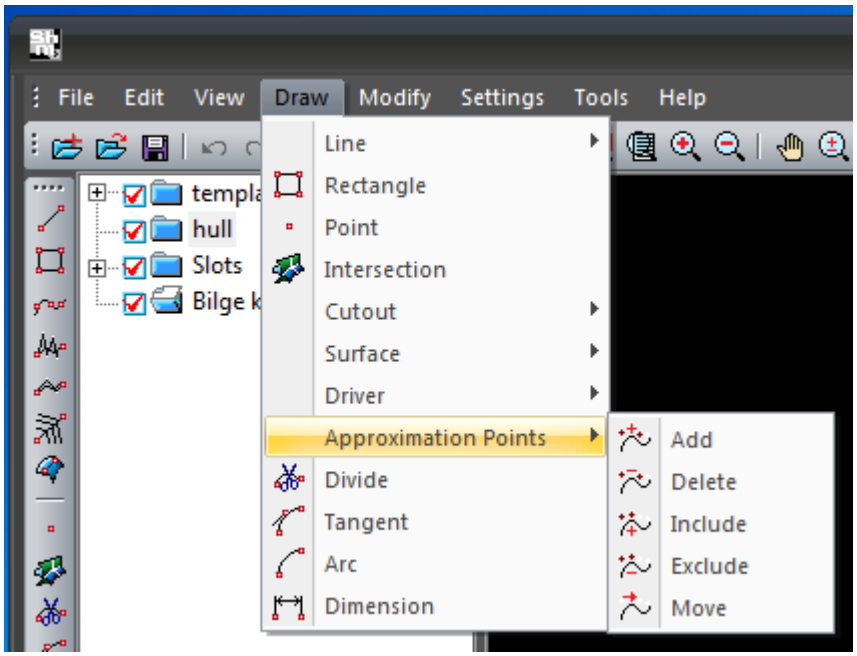

For method Driver select the driver to be exploded.

For method One Element select element of the driver to be exploded.

**Driver ▸ +/-**
This command allows adding a new element to the existing driver of similarity or removes its element.

Select the driver of similarity. All elements of the driver will temporarily be highlighted red. Select the elements. New elements will be added to the driver or driver elements to be removed from the driver of similarity.

## Draw ▸ Approximation Points




This menu contains commands that allow the approximation points to be assigned to and deleted from the lines or surfaces and to be further used for adjustment of the lines and surfaces. You can also adjust the approximation point coordinates.

Approximation points may also be loaded from the DXF-file or a special file.

### Approximation Points ▸ Add

Add command allows the new approximation points to be inserted for a specified line or surface.

Select the element. Input the approximation points by mouse of by coordinates. To stop the insertion press Esc or right-click the mouse.

### Approximation Points ▸ Delete

This command deletes the approximation points. Select the point. The approximation point will be removed from the project.

### Approximation Points ▸ Include

This command assigns approximation points with line of surface.

Select the element to which the approximation points will be assigned. Select approximation point.

### Approximation Points ▸ Exclude

This command excludes an approximation point from the line or surface.

Select the element from which the approximation points will be excluded. Select the approximation points.

### Approximation Points ▸ Move

Move the approximation points, which belong to an element.

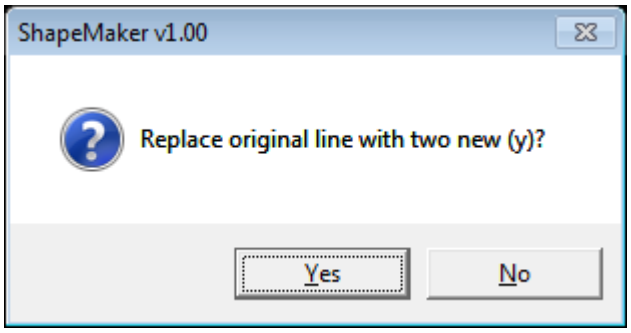
Specify the approximation points to be moved.

Select points to move and press Enter. If one point selected – move to new point position. If several points were selected follow instructions to move group of points.

## Draw ▸ Divide

This command is used to divide a line into two lines in the specified point.

Select the division point in the line. Select the required option in the following dialogue box:

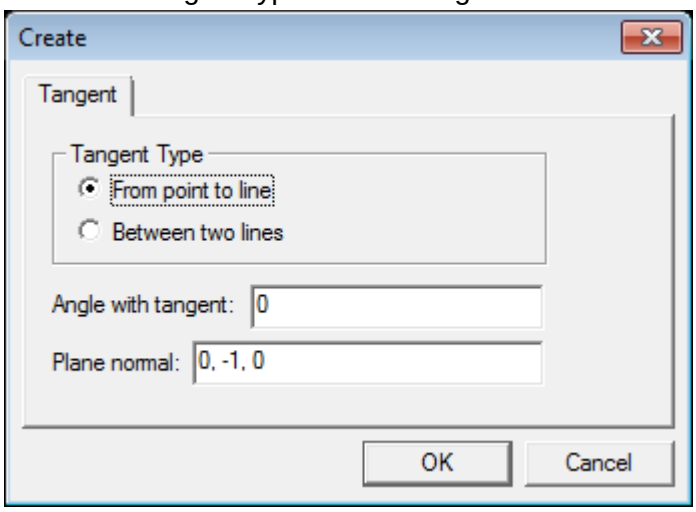



Two new lines will be created that exactly follow the shape of the source line. To stop the command press Esc or right-click the mouse.

## Draw ▸ Tangent

This command is used to create tangents to the lines.

Select the tangent type in the dialogue box.

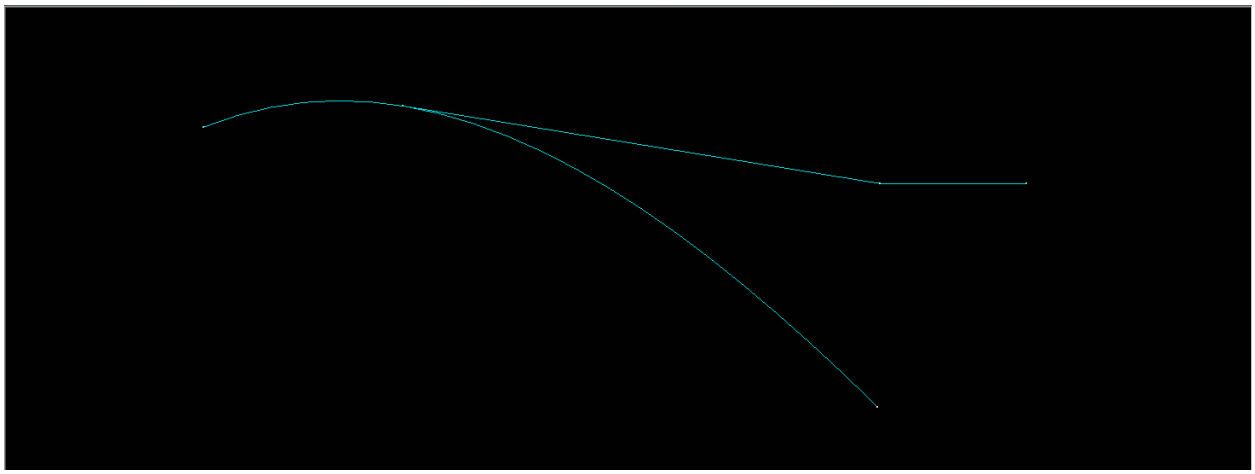



Angle with tangent – angle of slope to the tangent (for the type from point to line)

Plane normal – normal to projection.

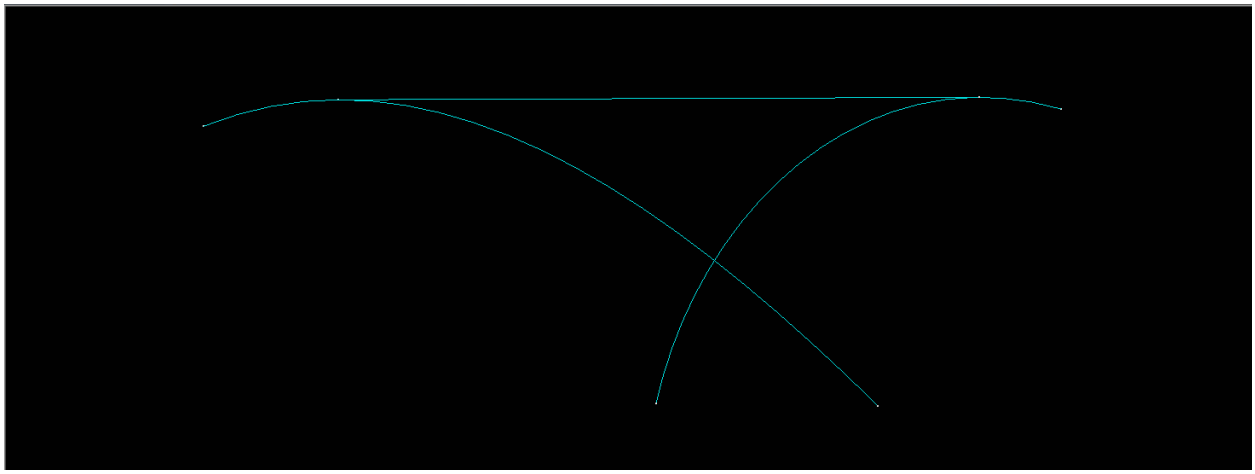
From point to line:

Define the first point of the tangent. Select line where you going to make tangent.



Between two lines:

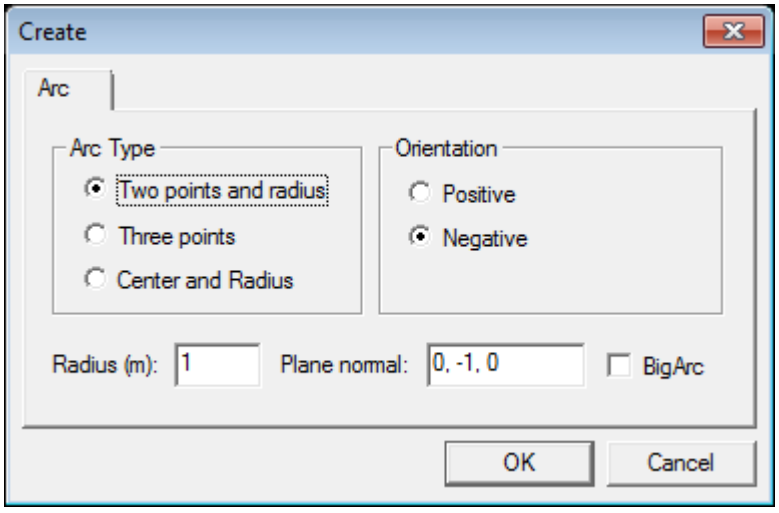
Select first line. Select second line.



## Draw ▸ Arc

This command is used to create arcs.

Select the arc type, radius and other required options.




There are 3 methods for creating an arc:

Two points and radius

Three points

Center and radius

Orientation: orientation of the arc.

Positive – counter-clockwise direction.

Negative – clockwise direction.

Radius – value of the arc radius, given in meters.

Plane normal – normal to projection plane.

BigArc – an arc with a smaller or larger central angle.

Two points and radius:

Select the second point of the arc. A new arc will be created. To stop input the arc press Esc or right-click the mouse.

Three points:

Select the first point. Select the second point of the arc. Select the third point of the arc. A new arc will be created. To stop input the arc press Esc or right-click the mouse.

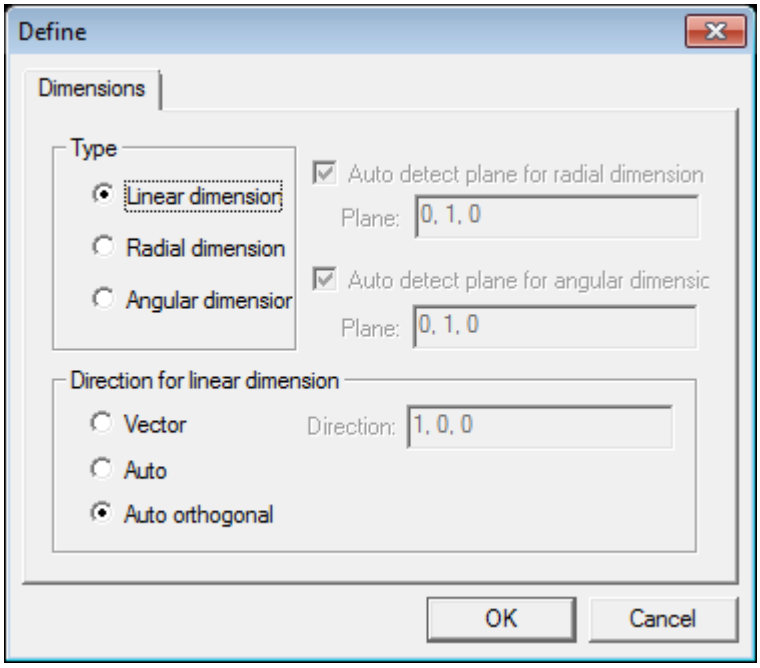
Center and radius:

Select center of the arc. Define the beginning angle of the arc. Define the end angle of the arc. A new arc will be created. To stop input the arc press Esc or right-click the mouse.

## Draw ▸ Dimension

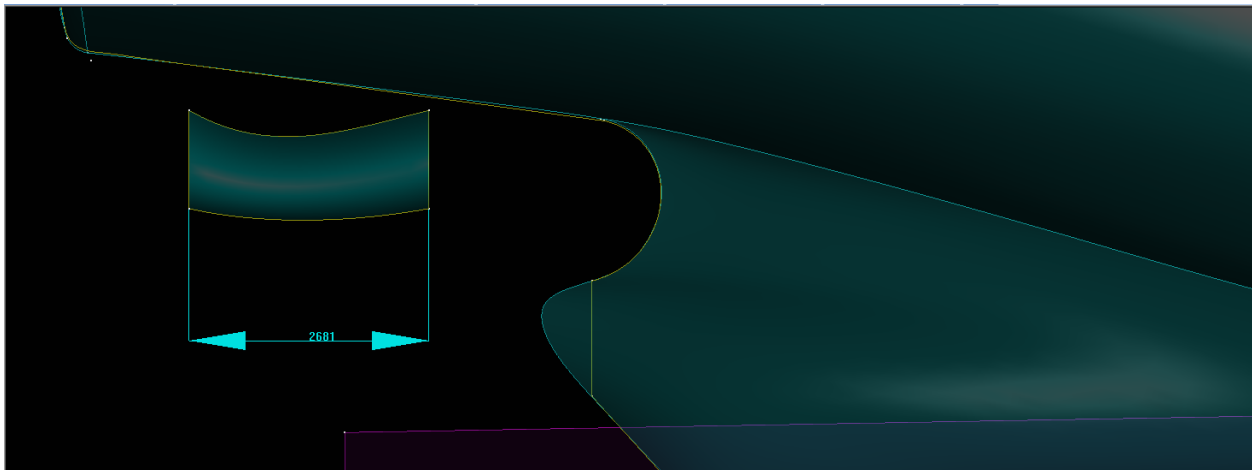
This command allows setting the linear, radial and angular dimensions.

Select type of the size and necessary parameters in the dialogue box.



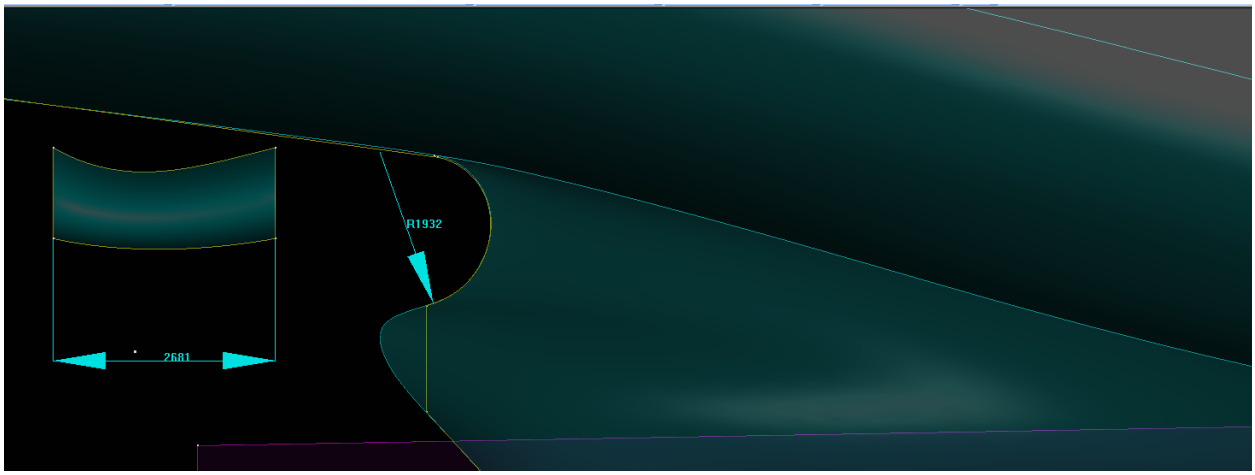

Linear dimension:

Select the first point of the linear dimension. Select the second point of the linear dimension. Select the text location point. The linear dimension will appear in the model.



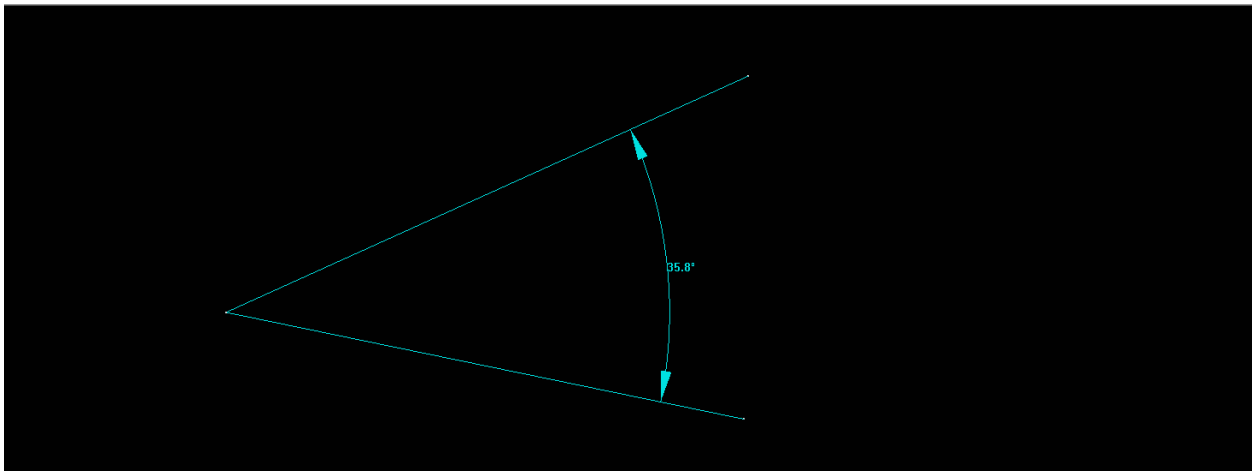

Radial dimension:

Select a point on the arc through which the dimensional line will pass. The specified radial dimension will appear.



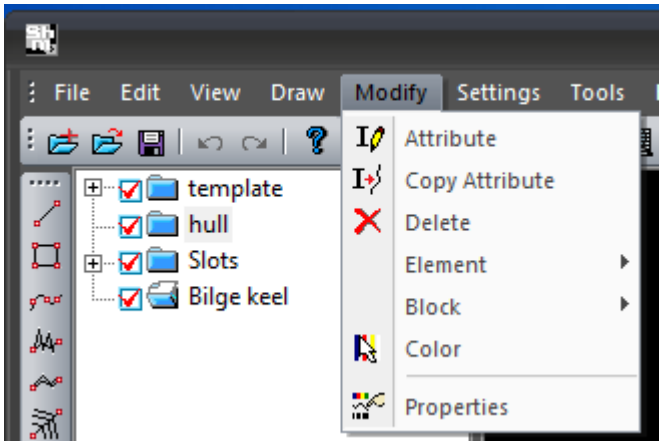

Angular dimension:

Select point of the first line. In this case the geometrical snap Point must be enabled. Select any point of the second line. In this case the geometrical snap Point must be enabled. The dimension with the text will be displayed on the screen.



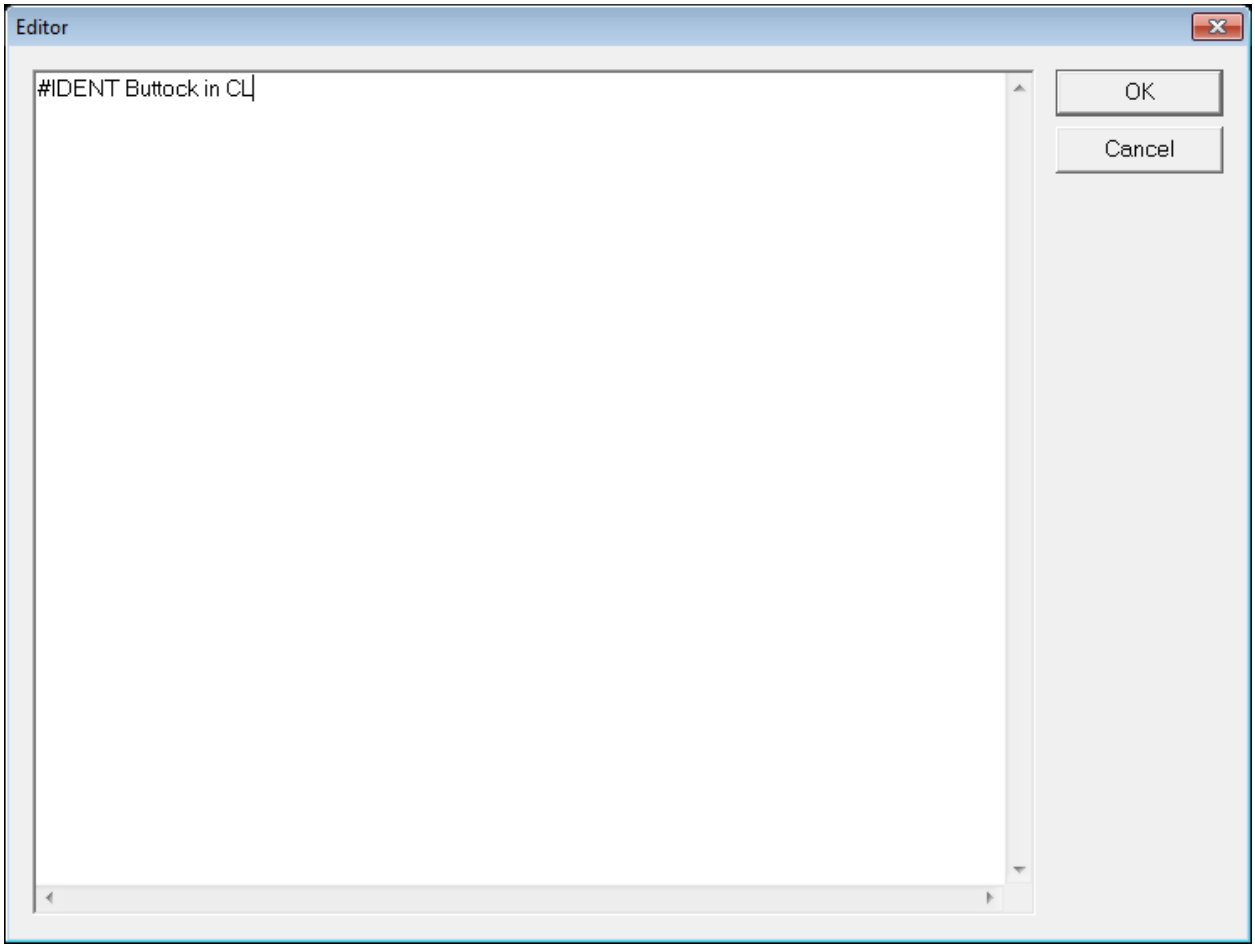
# Modify

Modify Menu contains commands that allow modify the geometrical characteristics of elements.




## Modify ▸ Attribute

This command is used for attachment of the different text attributes to elements.




## Modify ▸ Copy Attribute

This command is used for copy attached text attributes to other elements in the model.

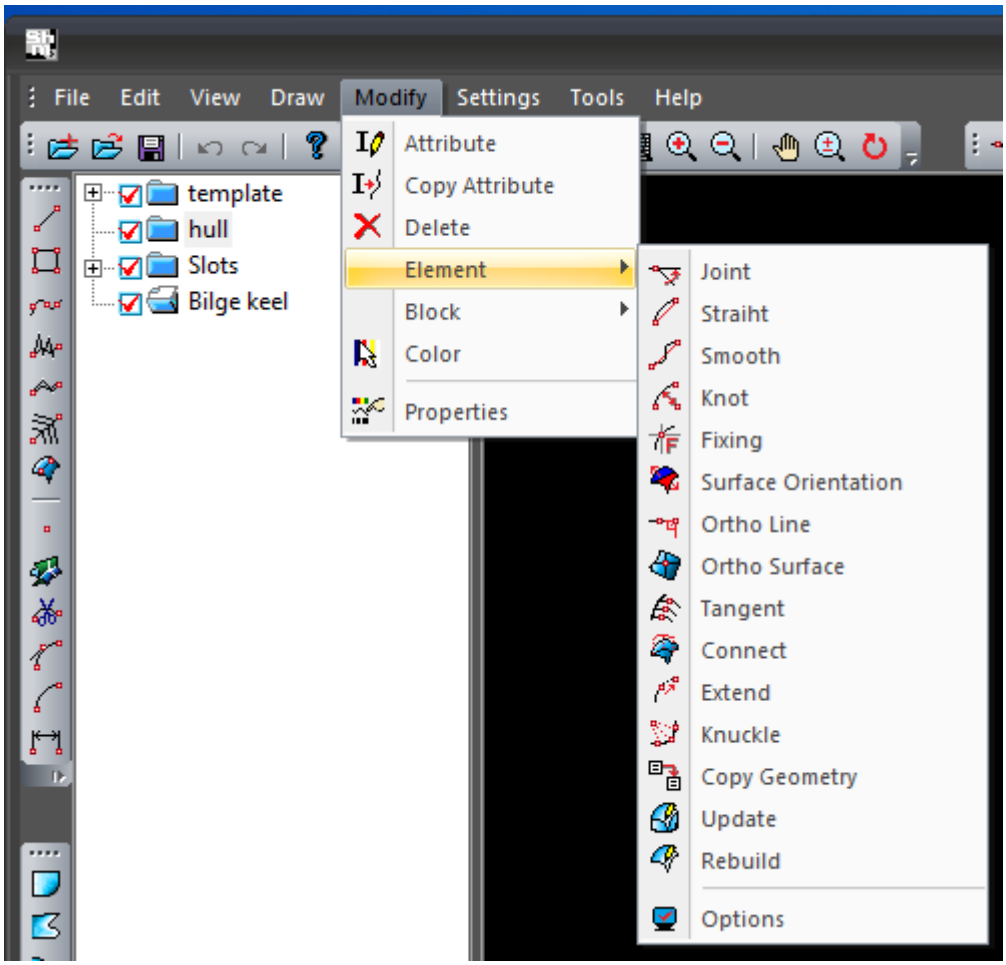
Select source element with the text information. Select element to copy text attributes. Edit text attributes in text editor if necessary.

## Modify ▸ Delete

This command deletes any element from the project that no other element has a reference to.

Select an element or elements to be deleted depending on the current method of selecting an element (see Selection Type).

## Modify ▸ Element

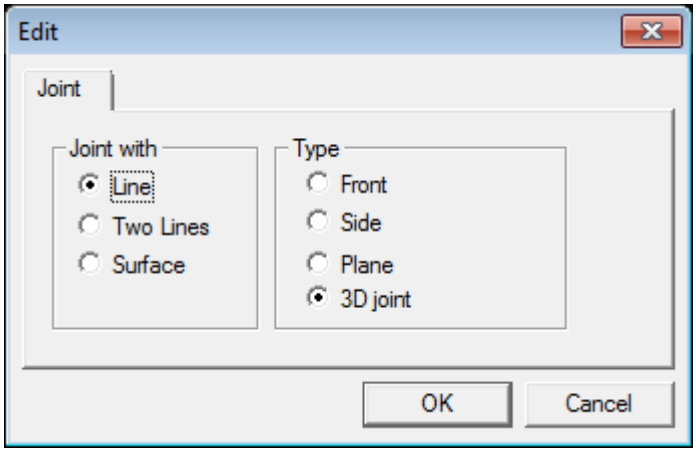



### Element ▸ Joint

This command is used to smoothly join a current edited line or a surface to another line or surface. Before use this command line or surface should be chosen for edit.

For line:

Select the type and parameters of joining.



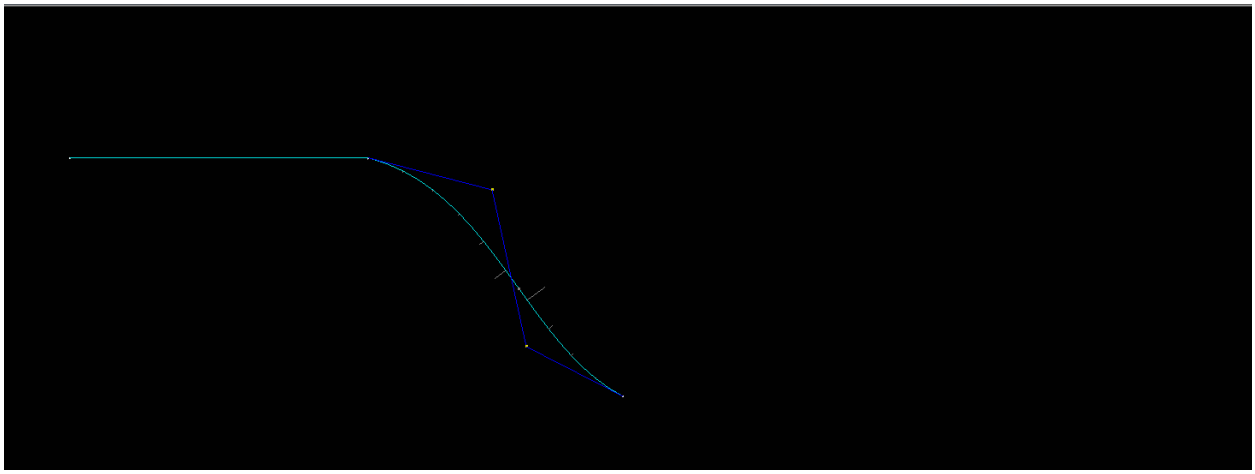

Joint with – select the element of joining.

Type – type of joining the elements: on projection Front**,** Side**,** Plane or in 3D joint.

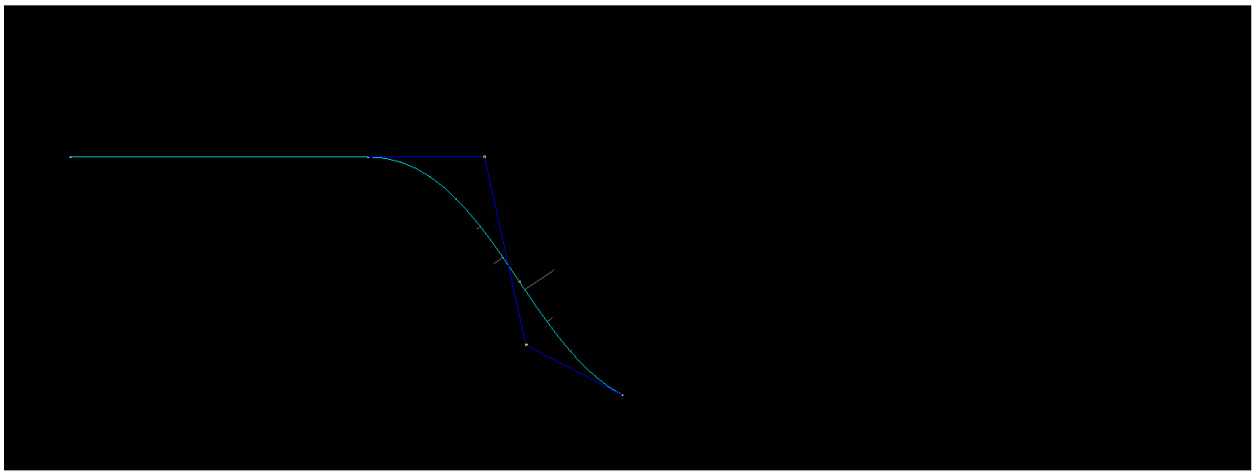
For method Line:

Modify control point of line closest to point of jointing.  Select a line to be jointed.

Before:



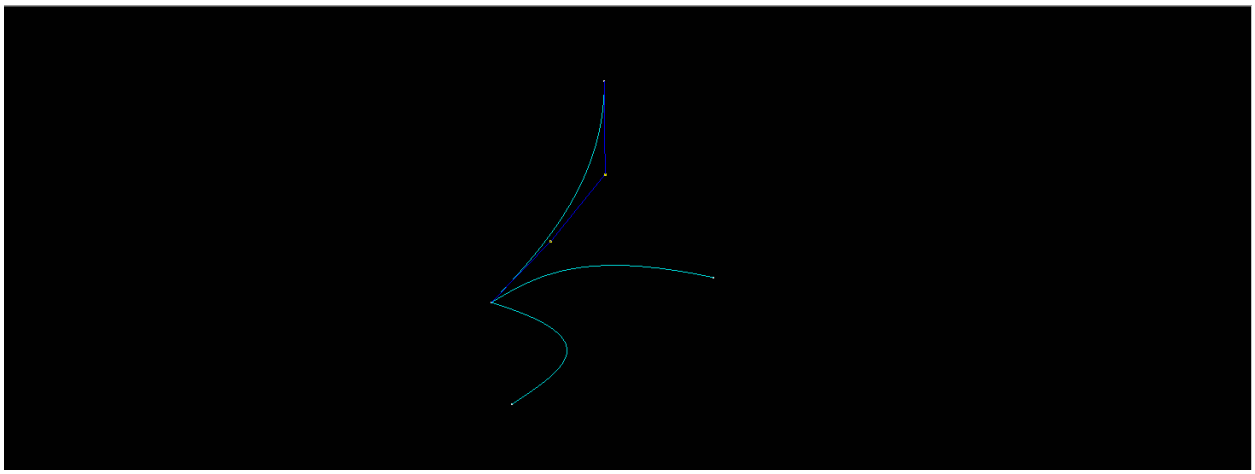
After:



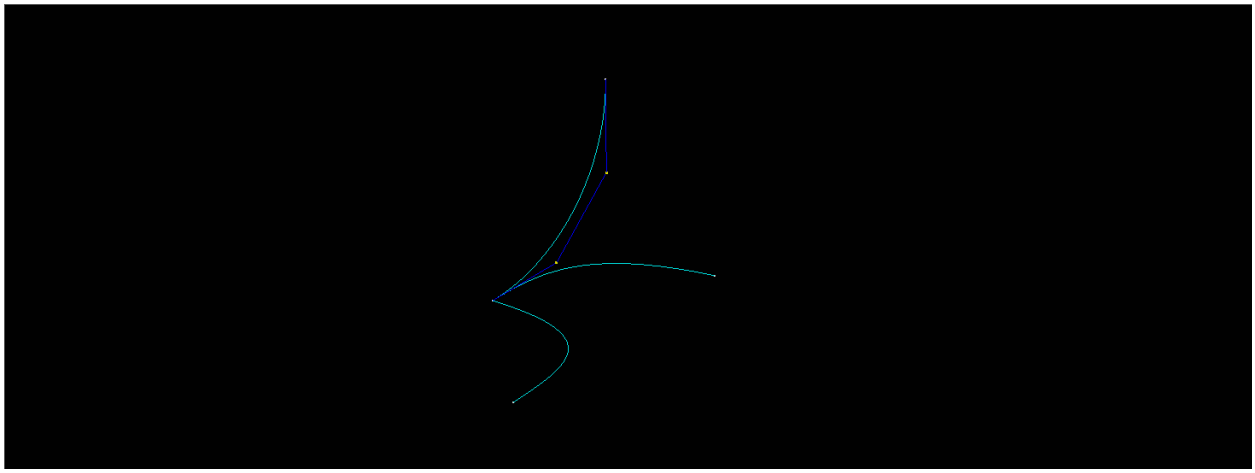
For method Two Lines:

Modify control point of line closest to point of jointing. Select two lines to be jointed with.

Before:



After:

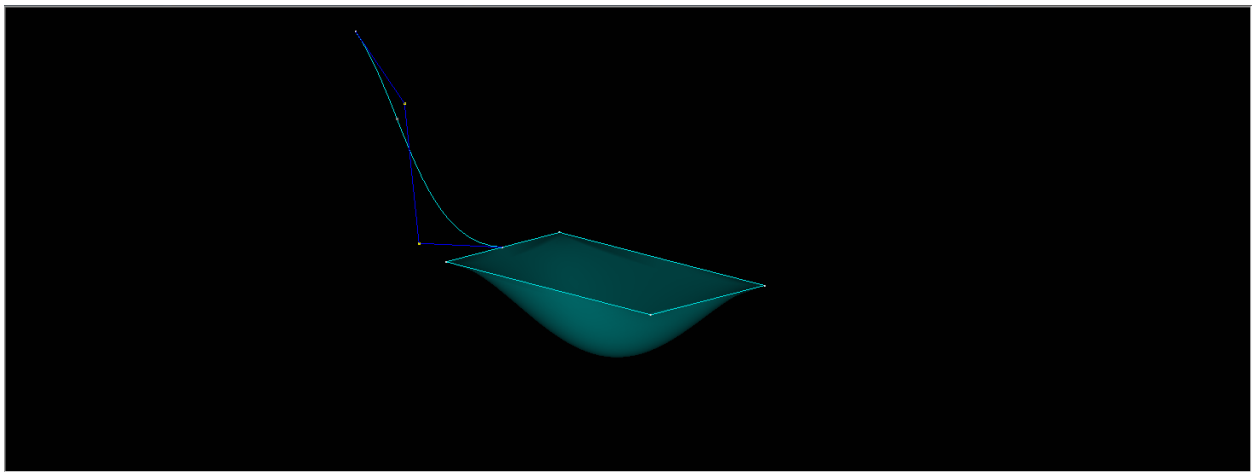


In this method of joining the tangent to a line in the end point will be lying in the plane determined by tangents to other two lines in the place of the joint.

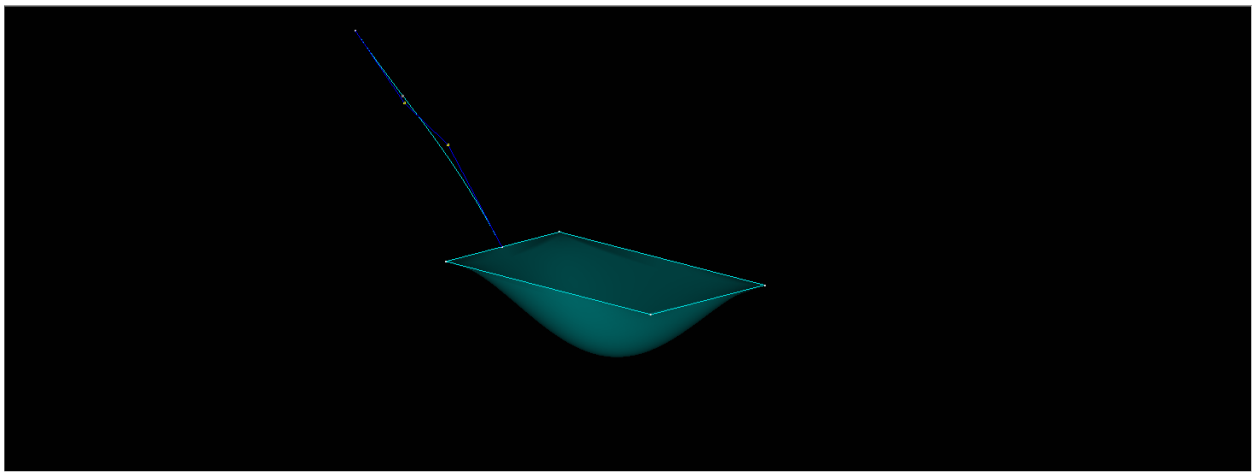
For method Surface:

Modify control point of line closest to point of jointing. Select a surface to be jointed.

Before:

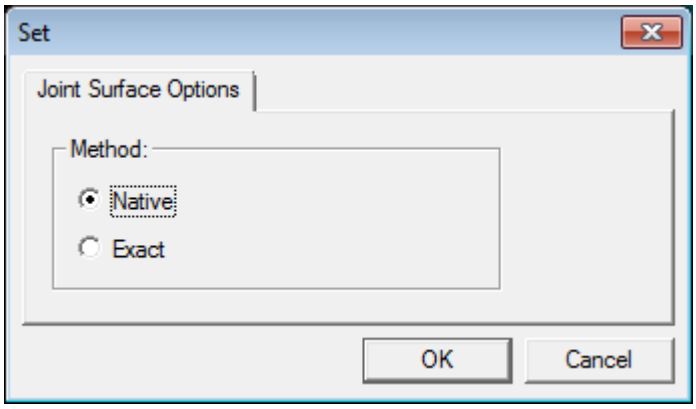


After:



For surface:

Select a surface to be jointed. Select type of joining in the dialogue box.

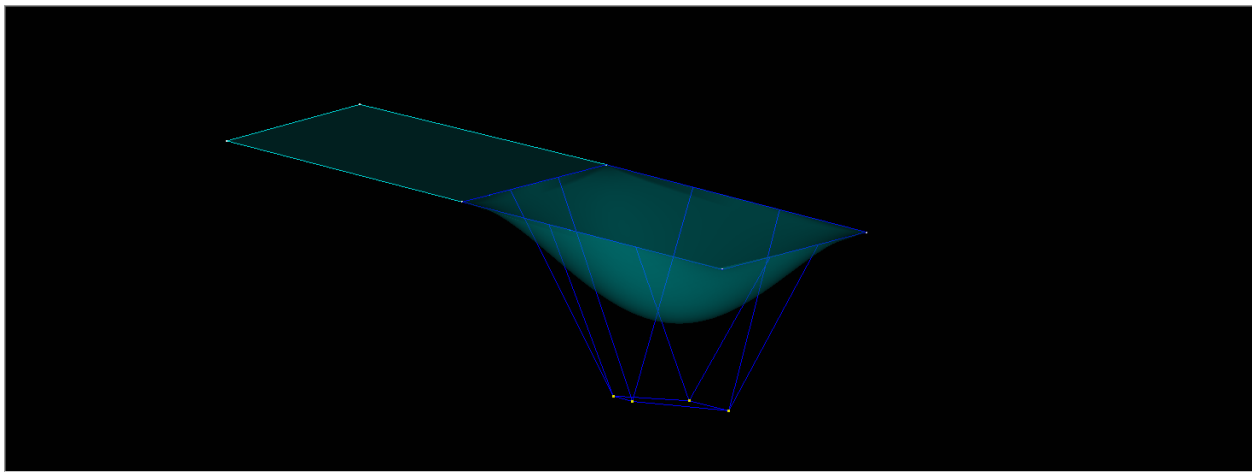



Native – ensures contact of surfaces in the boundary line nodes (B-spline nodes). The surfaces may not be jointed along the whole border.

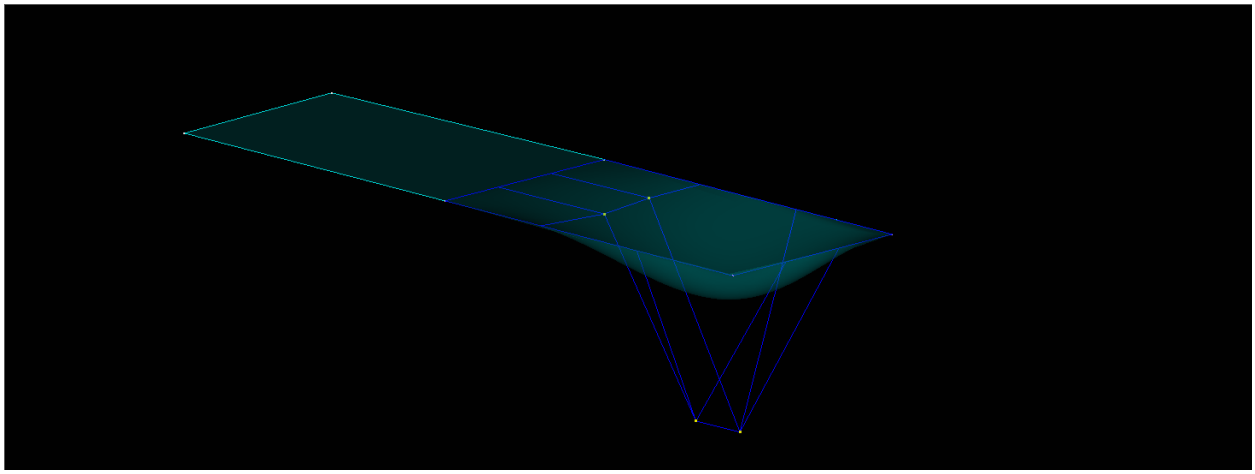
Exact – the surfaces will be smoothly jointed along the whole border.

Select joining line of two surfaces. Select jointing surface.

Before:



After:

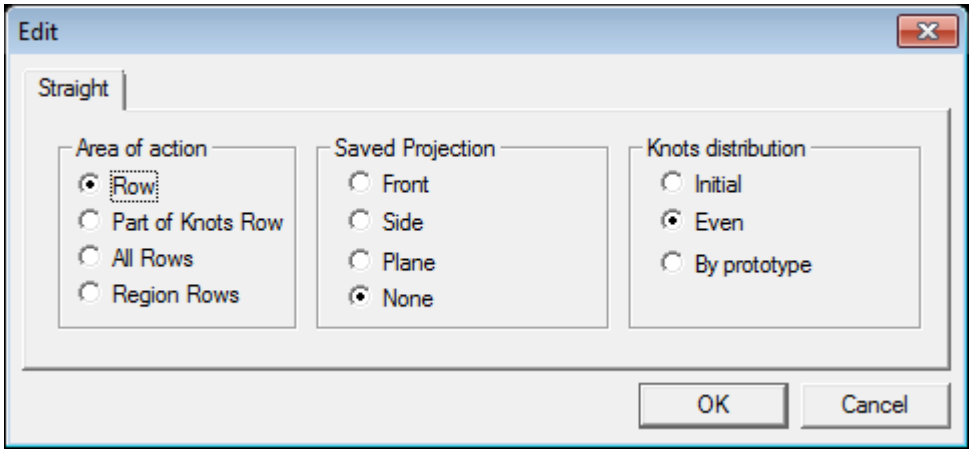


**Element ▸ Straight**

This command straightens a part of or the whole control polygon (in space or retaining one of projections) or a series of control points of the surface.

This will result in straightening of the relevant part of the line or surface.

Select an element to be edited. Select straightening method in the dialogue box.




The following straightening options and modes are available:

Area of action:

Row - straightening a whole line (in case of surface – selected knots row).

Part of Knots Row – a part of the control polygon between two knots will be straightened.

All Rows – straightening all surface knots rows (in case of line – it works like Row).

Region Rows - straightening all surface knots rows in selected region.

Saved Projection: - projection to be saved:

Front**,** Side**,** Plane – saving of selected line projection.

None – straightening in 3D;

Knots distribution - distribution of knots after straightening:

Initial – original,

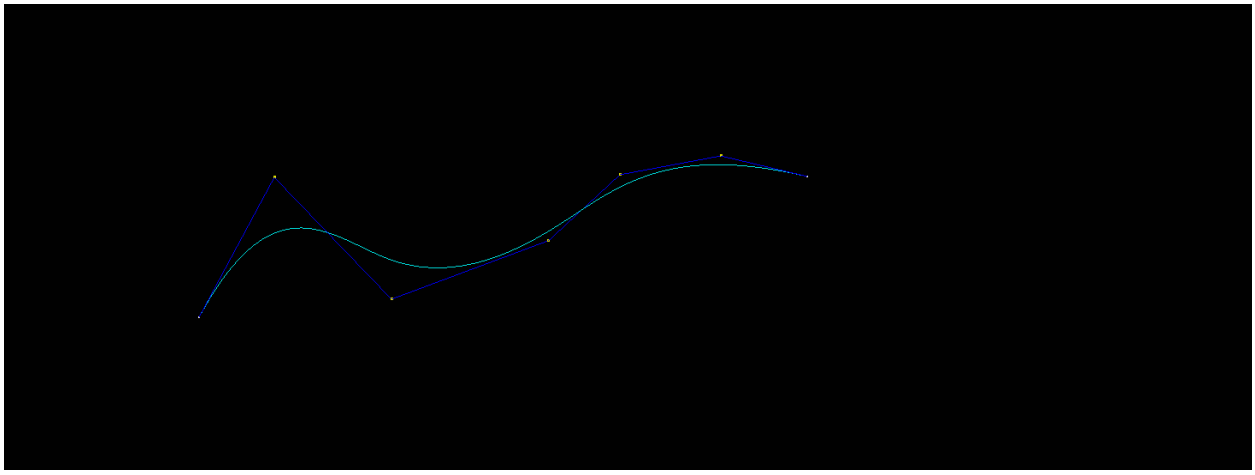
Even – equal distance between control points,

By prototype – by the knots distribution prototype on the other element.

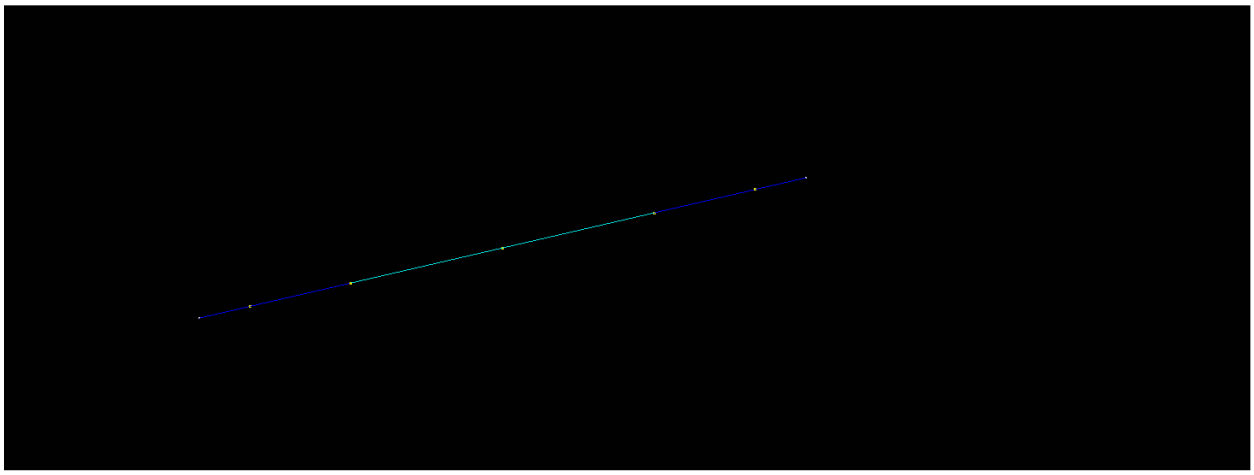
For line:

Row:

Before:



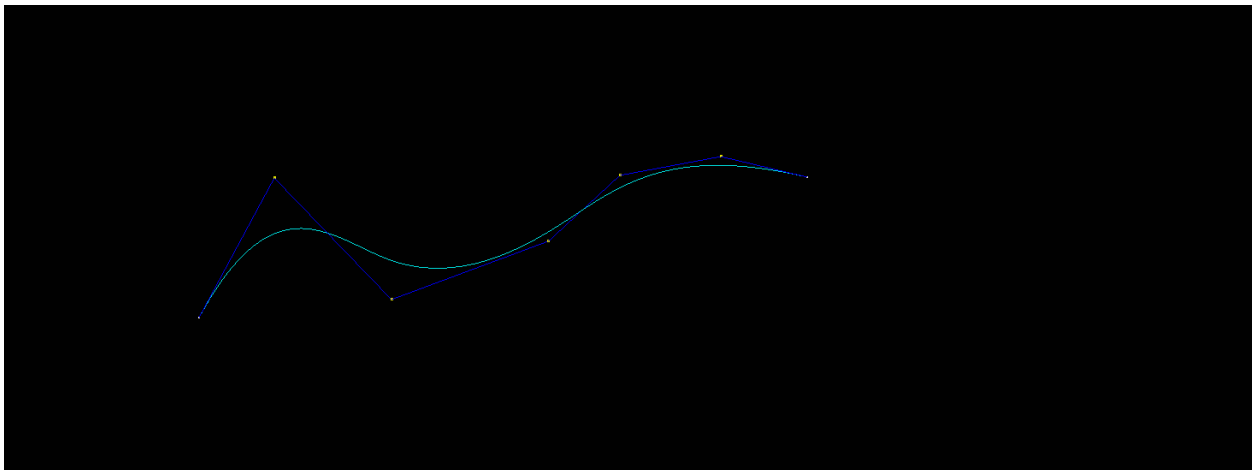
After:



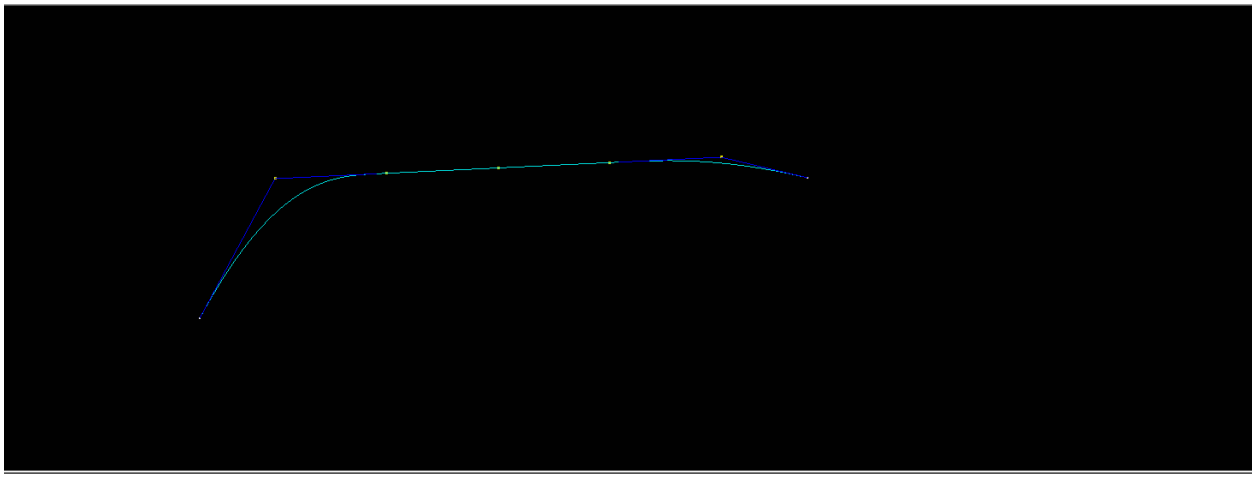
Part of Knot Row:

Select begin control point/ Select end control point.

Before:



After:

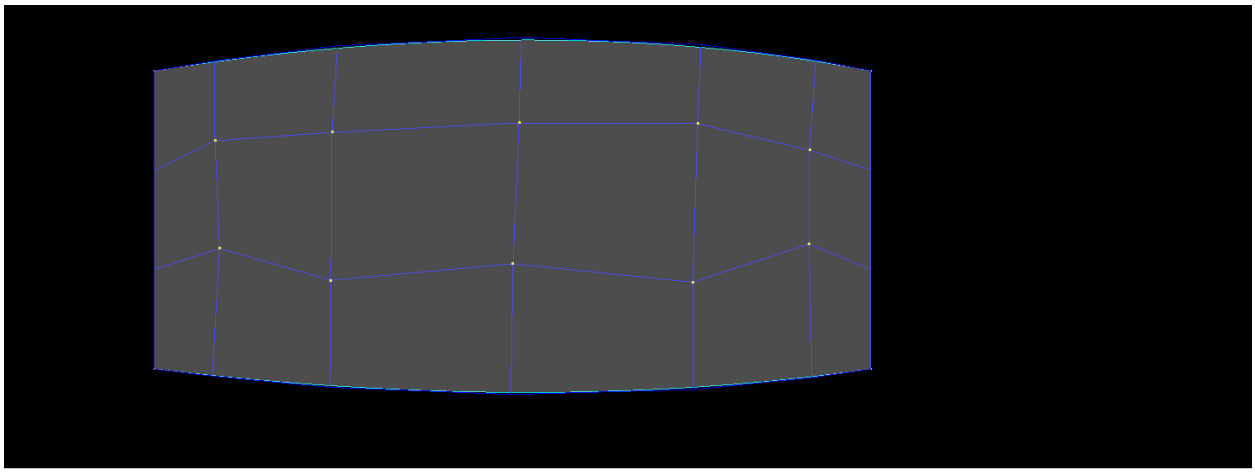


For surface:

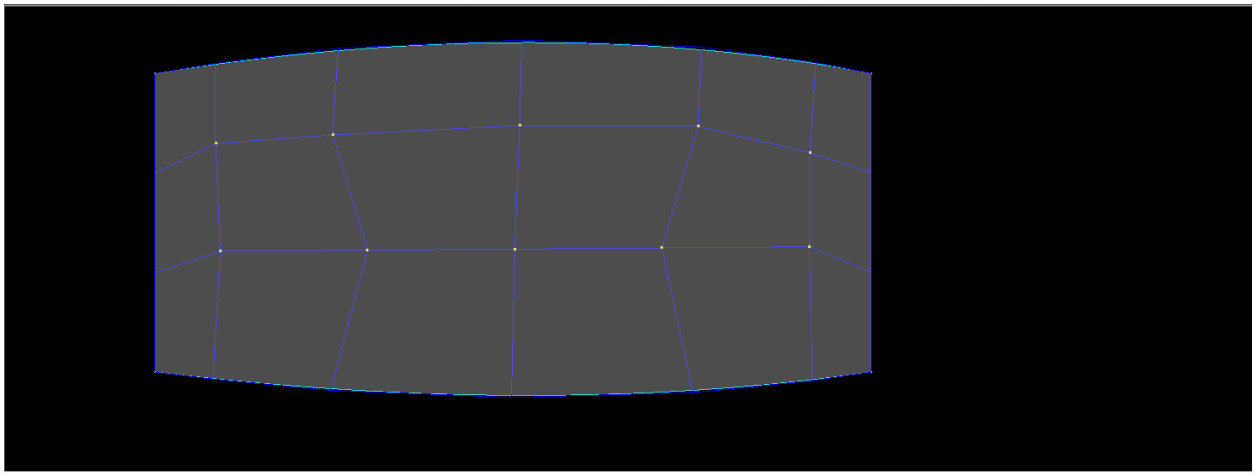
Part of Knots Row:

Select begin control point. Select end control point.

Before:



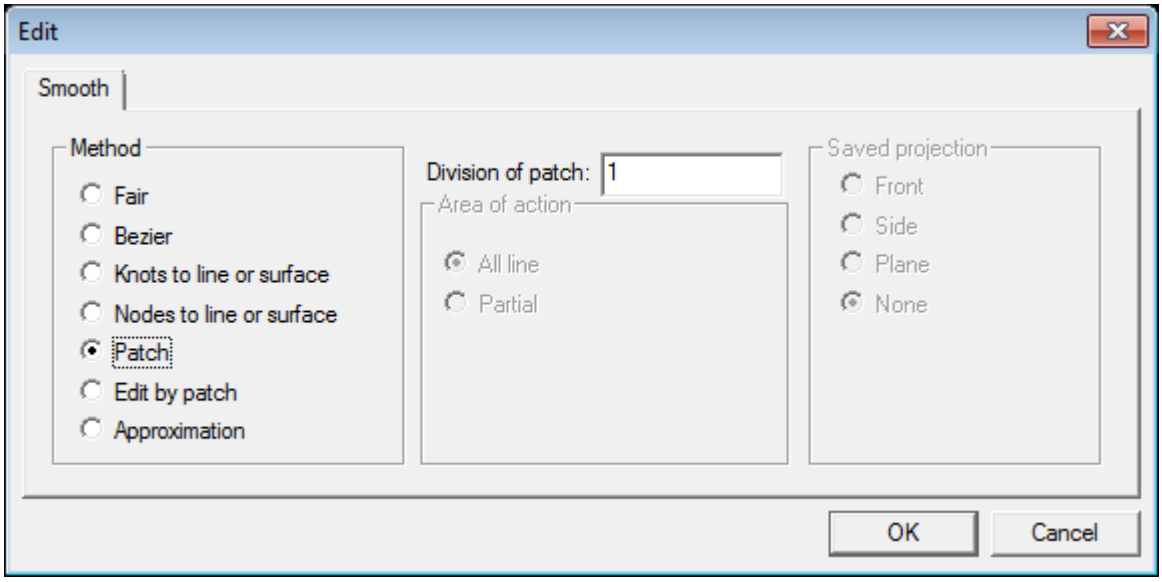
After:



The area of knots will be straightened according to the selected method.

**Element ▸ Smooth**

This command allows smoothing lines and surfaces. Select an element to be edited. In the dialogue box select smoothing mode and options for a line.




Method:

Fair – smoothing consists in projecting the knots on the line itself. The method is effective in case of a large number of knots and inconsiderable dispersion;

Bezier – the knots are laid on a special Bezier line defined or edited during the smoothing process.

Knots to line or surface – knots of the specified area of the control polygon are laid on another (previously specified) line or surface specified by the user on the system request.

Nodes to line or surface – nodes of the specified area of the line are laid on another (previously specified) line or surface specified by the user on the system request.

Patch – the knots are laid on a special Bezier surface patch.

Edit by patch – the knots are laid on a special Bezier surface patch edited during the smoothing process.

Approximation – adjustment the curve to approximation points.

Area of action: - area of action during the smoothing process.

All line – area of action is the whole line;

Partial - area of action is a part of the line;

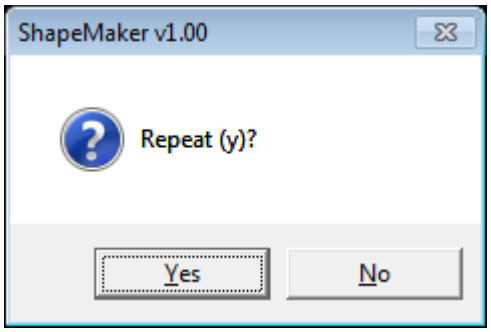
Saved projection:

Front**,** Side**,** Plan - saved projection during the smoothing process;

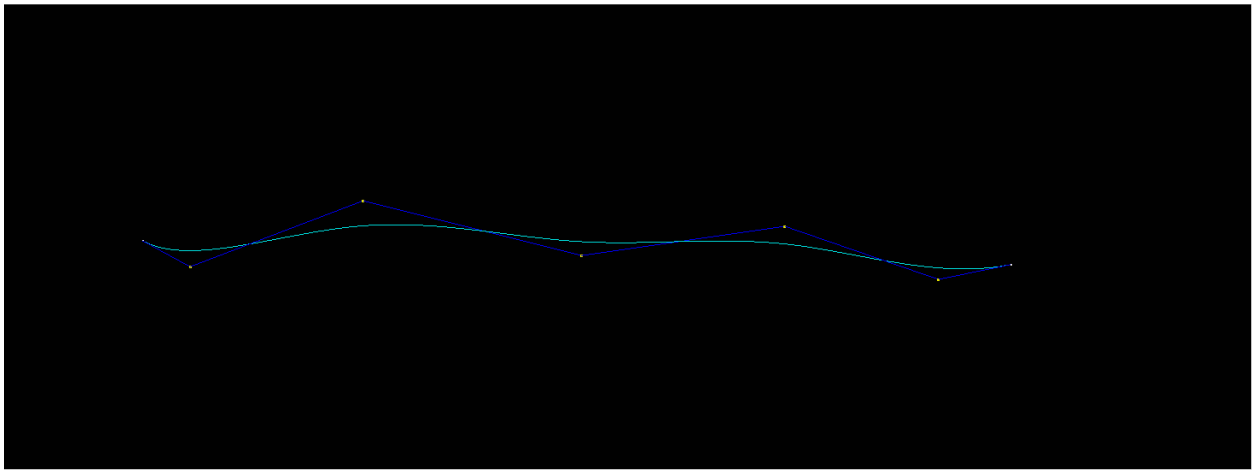
None – without saving in any projection;

Fair:

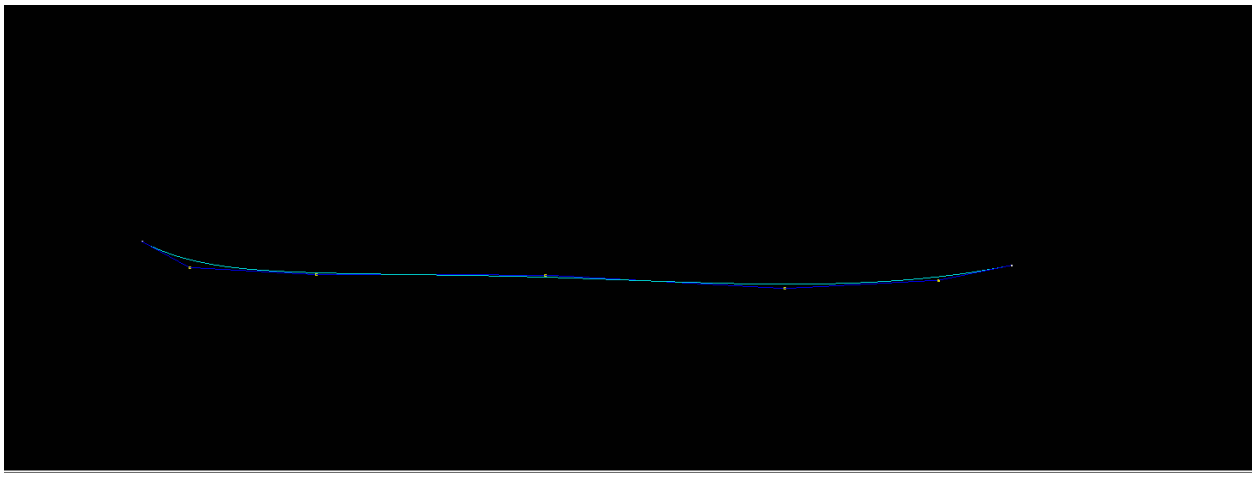
Select beginning point/ Select end point. In the dialogue box click Yes to continue. To stop the process of smoothing click No.




Before:

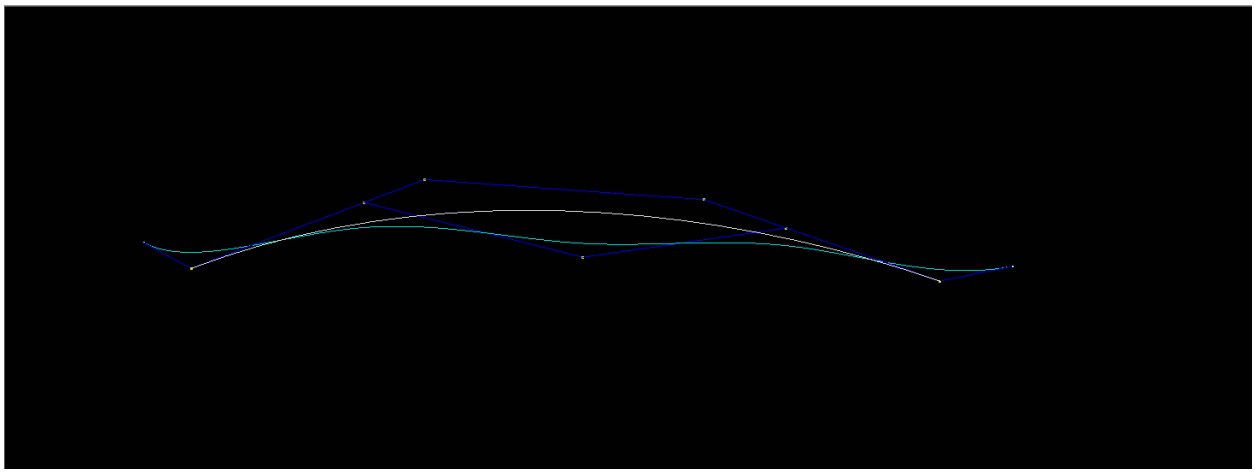


After:

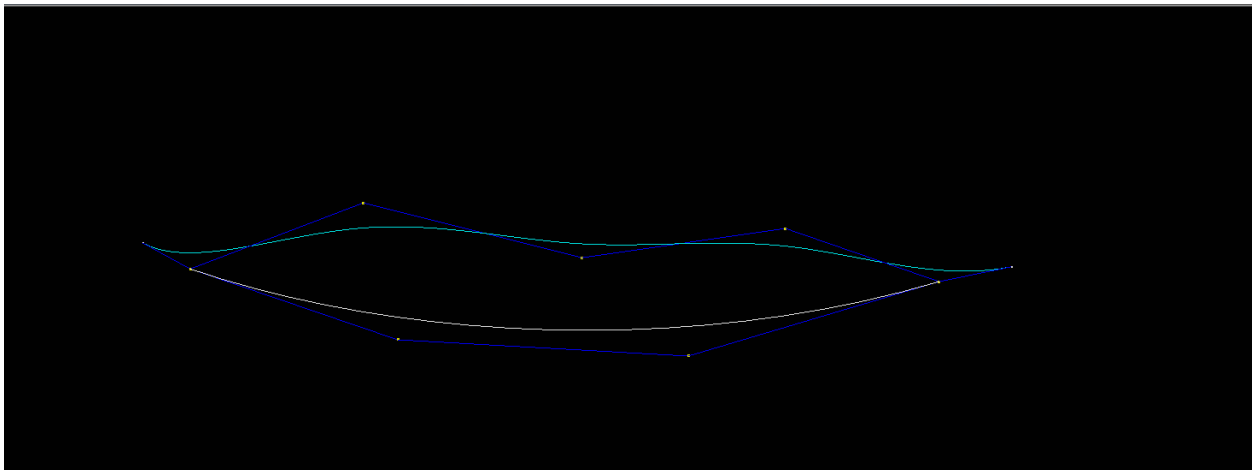


Bezier:

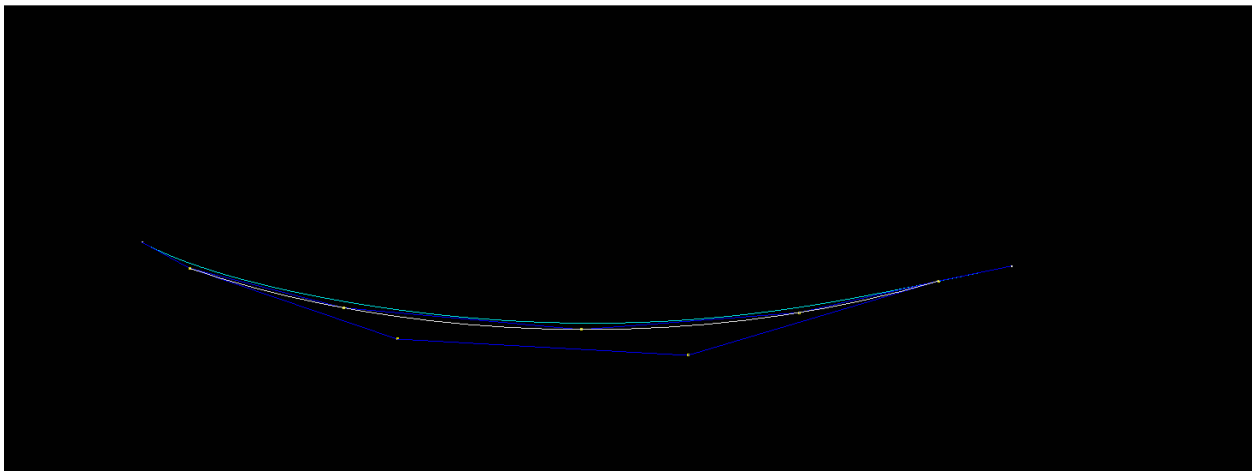
Select beginning point. Select end point.  Bezier curve will be created on top of the selected area.



Modify position of the Bezier control polygon.



Press Esc after finish the modification. All control point of the curves selected area will jump to Bezier curve.

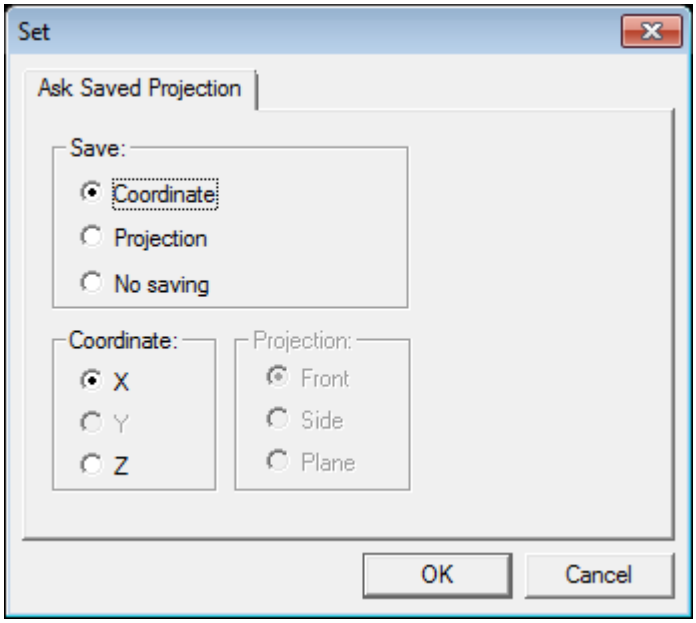


Press Esc to exit the smoothing mode and continue correcting the smoothing curve.

Knots to line or surface:

Select beginning point. Select end point**.** Select line or surface.

In the dialogue box select the knots distribution options:




Save:

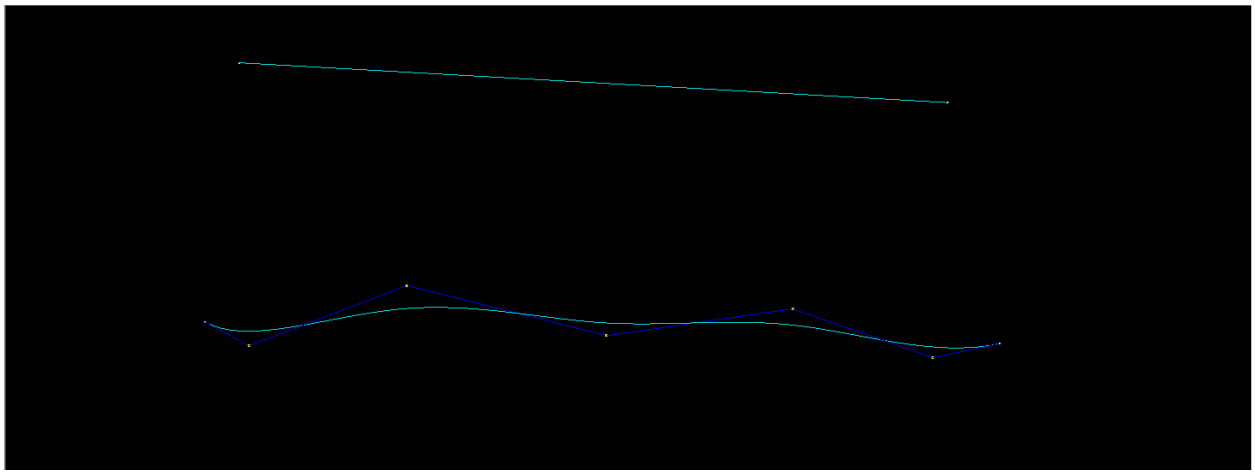
Coordinate – according to selected Coordinate

Projection – according to selected Projection

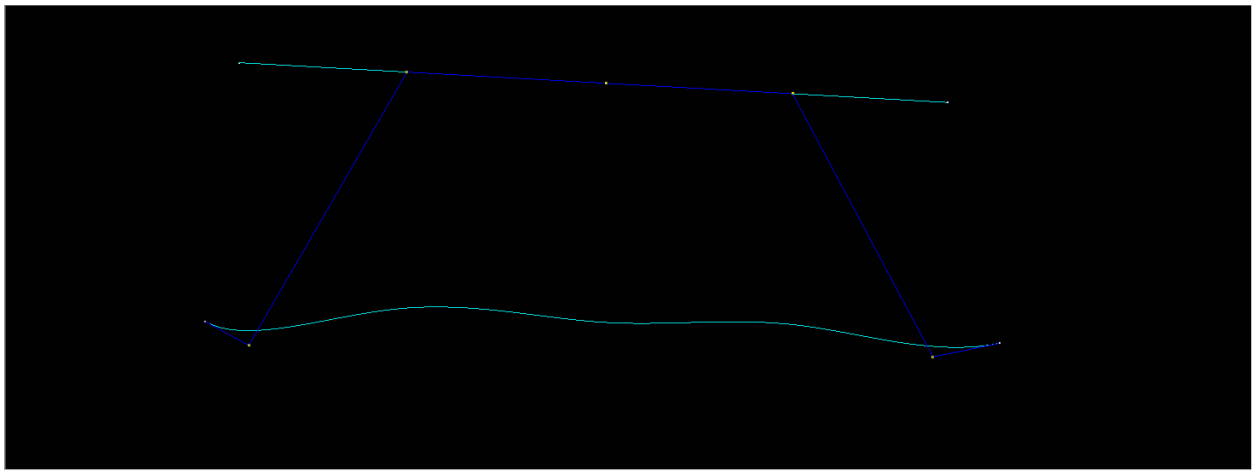
No saving – by shortest distance to line or surface

The selected area of the line will be smoothed in accordance with the selected line.

Before:



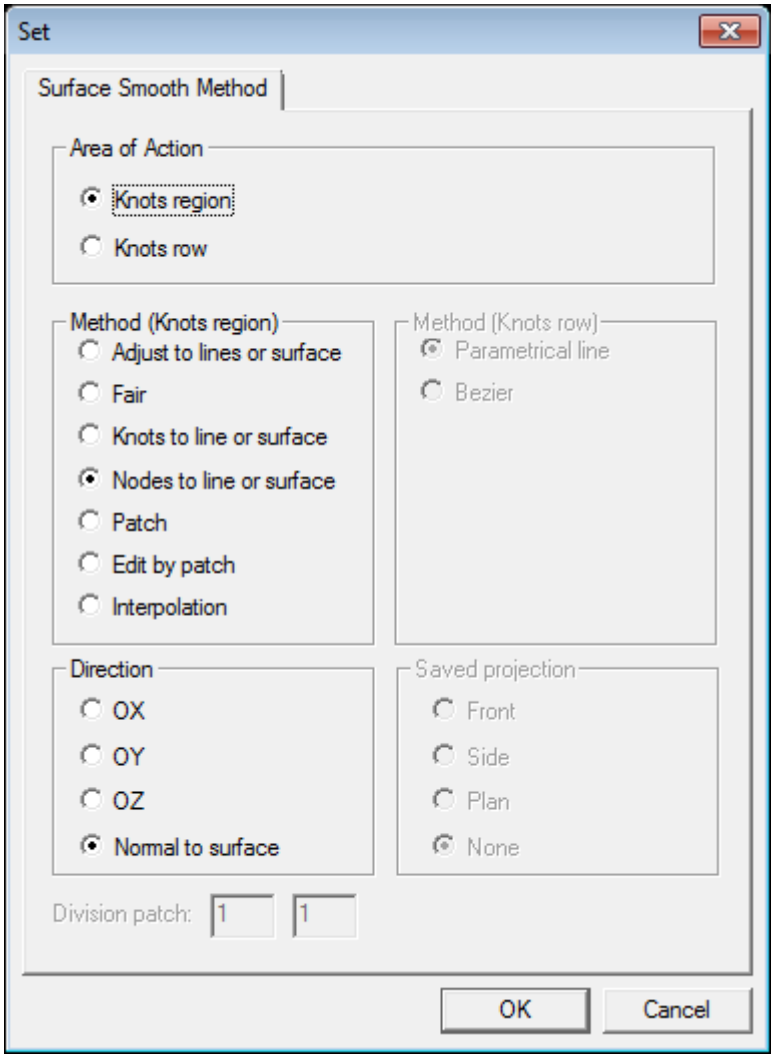
After:



For method Nodes to line or surface: - see method Knots to line or surface.

For method Patch and Edit by Patch: - see method Bezier (difference is only in number of patch control points).

Select a surface to be edited. In the dialogue box select smoothing mode and options for a surface.




Area of Action: - area to be smoothed.

Knots region – smoothing of the knots area;

Knots row – smoothing a series of knots.

Method (Knots region): - method of smoothing the knots area:

Put on lines – approximation a set of 3D lines net of surfaces;

Fair – surface curvature smoothing.

Knots to line or surface – knots of the specified area are laid on another line or surface.

Nodes to line or surface – nodes of the specified area of are laid on another line or surface.

Patch - knots of the specified area are re-interpolated according to the point's distribution law of the given area of the B-spline surface;

Edit by patch - knots of the specified area are re-interpolated according to the point's distribution law of the given area of the B-spline surface and the user taking control over the reference points of the specified area of the surface changes position of the knots of the edited surface.

Method (Knots row): - a method of smoothing a series of knots:

Parametrical line - smoothing consists in projecting the knots on the line itself.

Bezier – the knots are laid on a special Bezier line defined or corrected during the smoothing process;

Direction: - direction in which the knots or nodes of the surface are moved.

OX – in direction of X coordinate;

OY – in direction of Y coordinate;

OZ – in direction of Z coordinate;

Normal to surface – in direction of a normal to surface.

Saved projection: - saved projection during the process of smoothing.

None – without saving in any projection;

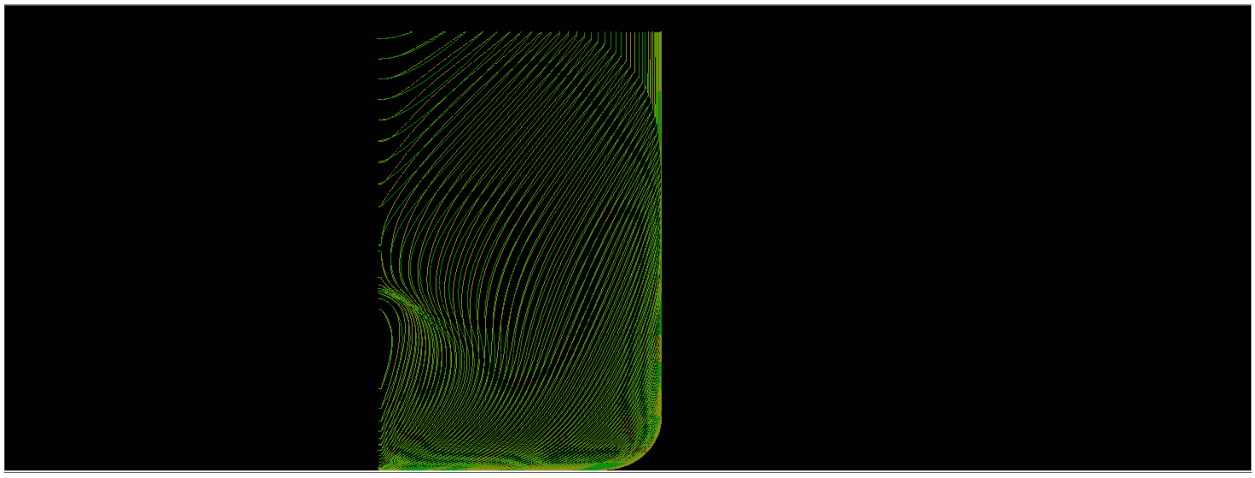
Division patch: - temporary subdivision of the edited surface area into the required number of nodes.

Put on lines:

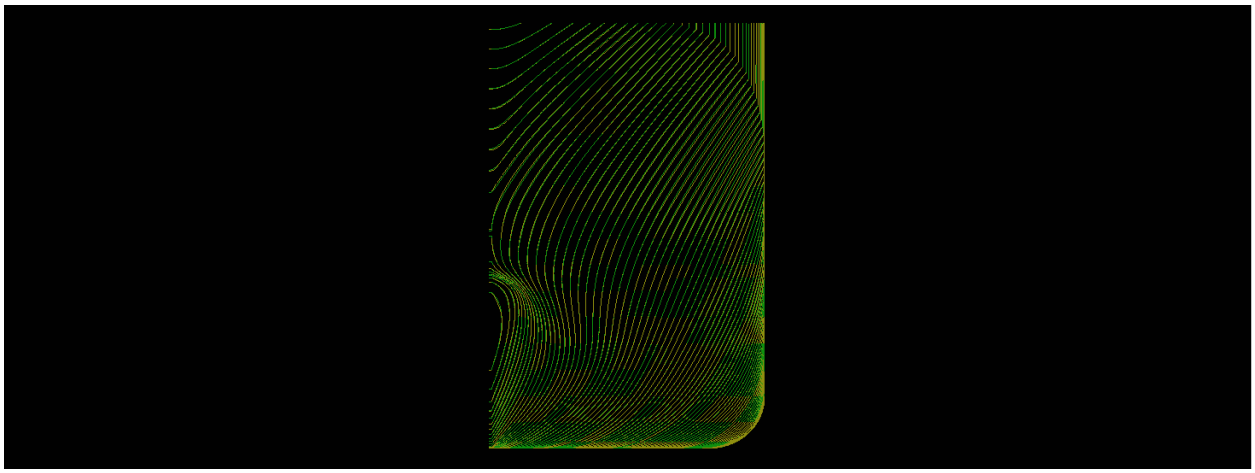
This method allows approximation a set of 3D lines net or surfaces.

Select row knots - specify the main direction of approximation. Select Elements and Press Enter - select set of 3D lines or surfaces. Select begin point of the control point’s area. Select end point of the control point’s area.

Before:



After

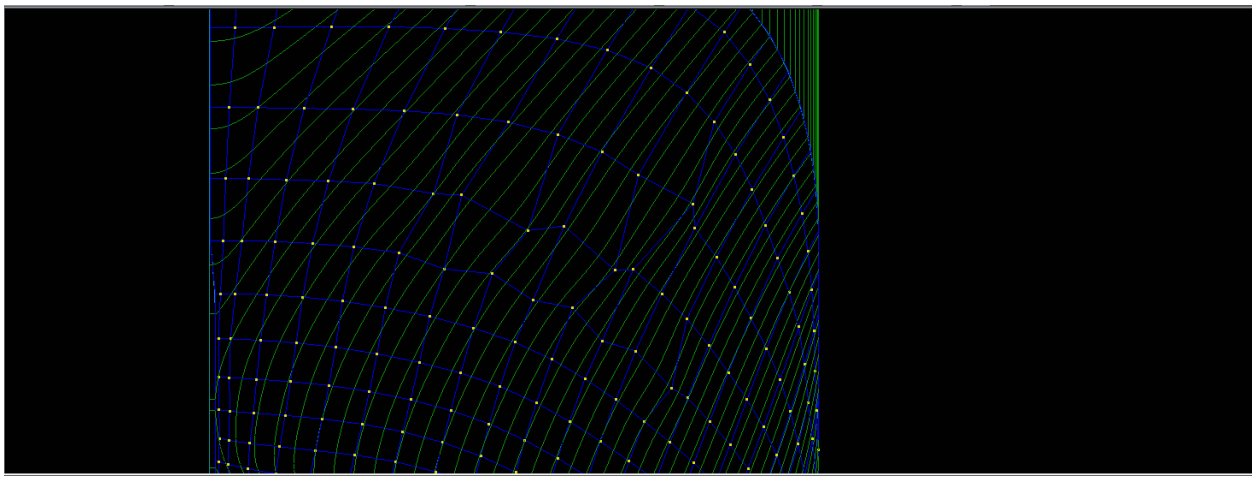


Fair:

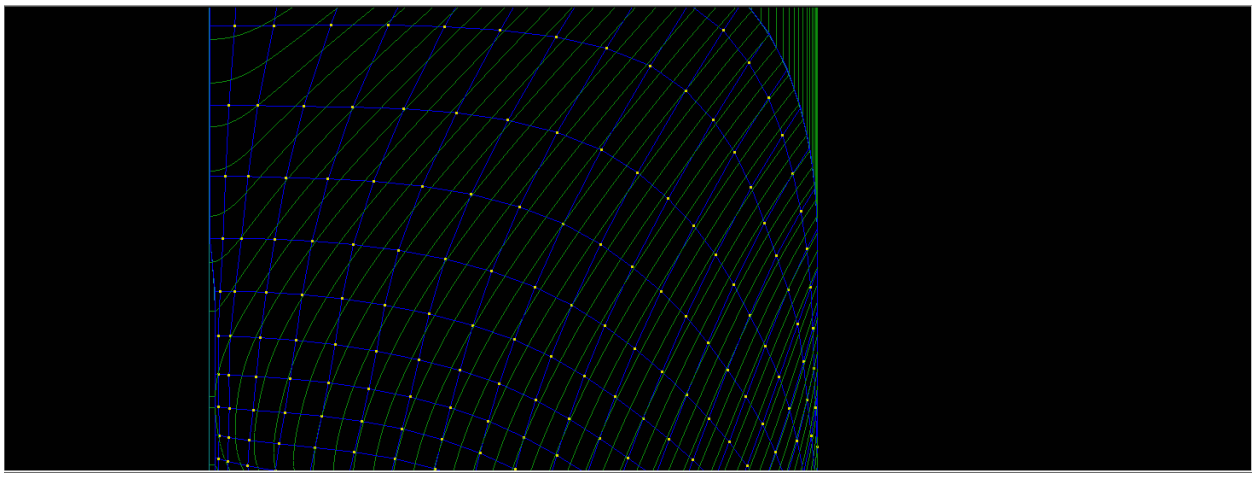
This method allows surface curvature smoothing.

Select begin point. Select end point. To complete the operation press Esc.

Before:



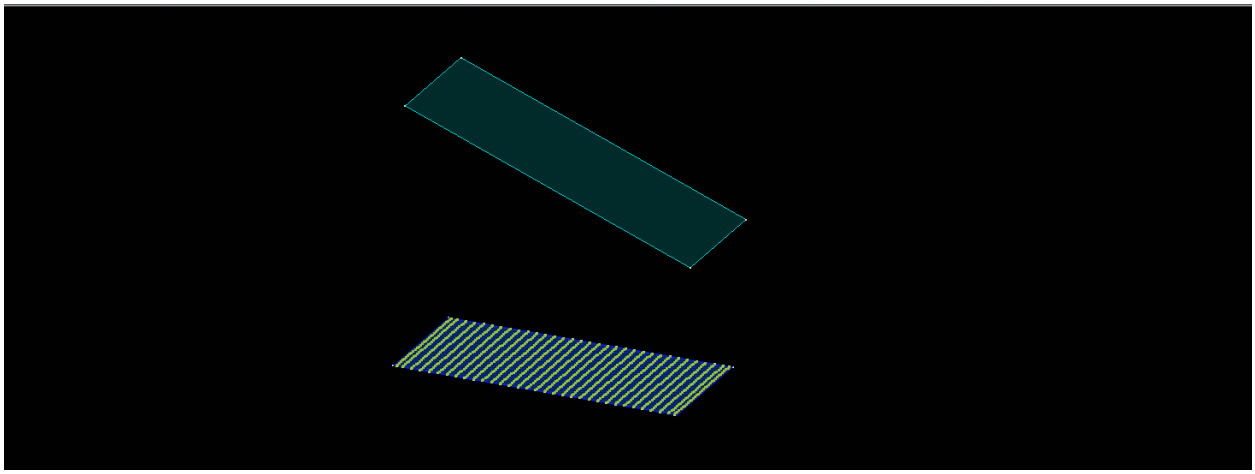
After:



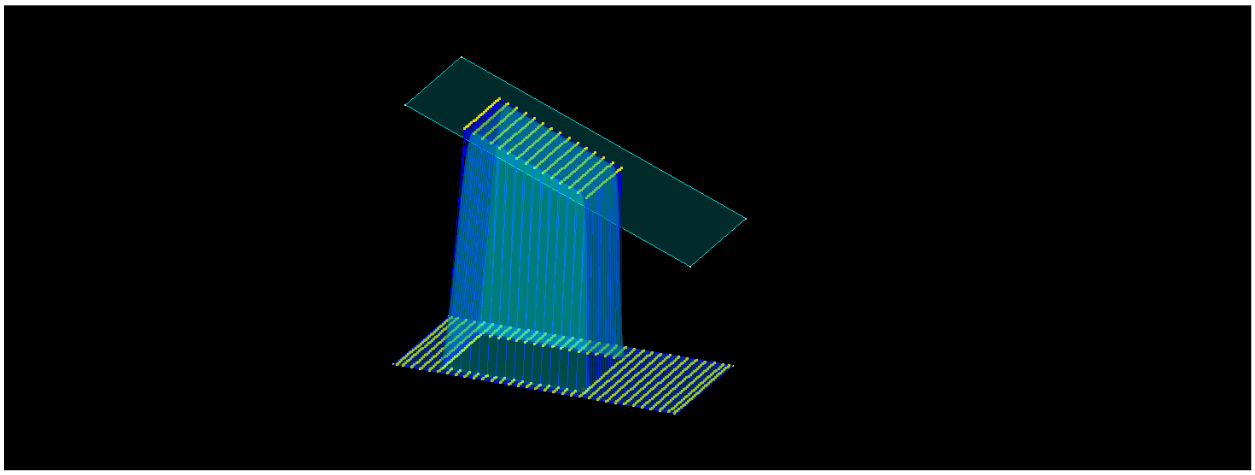
Knots to line or surface:

For method Knots to line or surface with the area of action on surface, if you want to smooth a series of knots take the actions similar to those taken to smooth the lines by this method. If you want to smooth an area of knots, on selecting the area the following request will be displayed: "Select surface:". Specify a surface on which the knots are to be laid.

Before:



After:



Nodes to line or surface:

Method Nodes to line or surface are similar to knots to line or surface method, but in this method all manipulations have to do with nodes instead of knots.

Patch and Edit by patch:

Both methods are similar to same methods for lines, but all manipulation doing with region of surface knots.

Interpolation:

Interpolation selected area of control points as superposition of boundary lines.

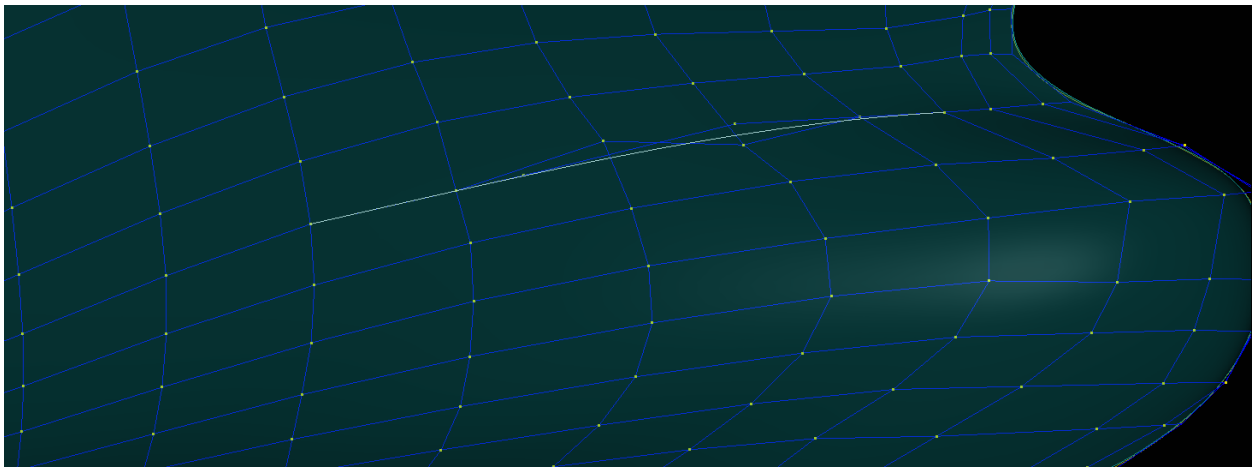
Parametrical line:

Method parametrical line is completely similar to the method fair for a line.

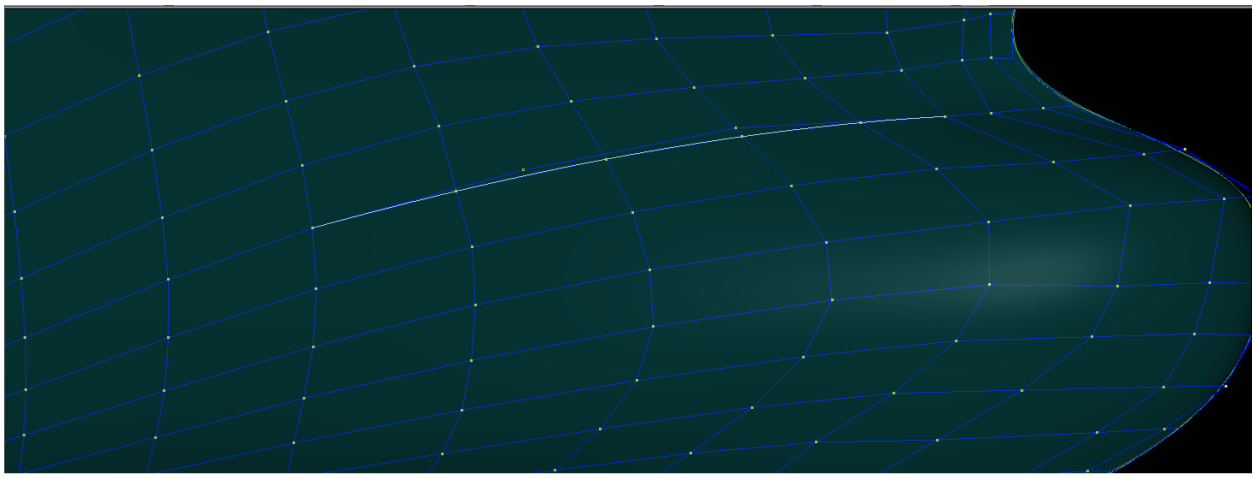
Bezier:

Method Bezier – the knots are laid on a special Bezier line defined or corrected during the process of smoothing similarly to smoothing the line by this method.

Before:



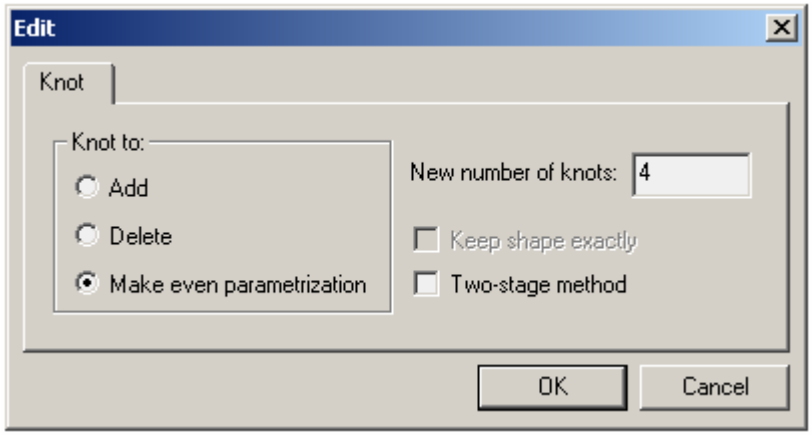
After:



### Element ▸ Smooth

This command is use to control number of control points b-spline.

Select an element to be edited. In the dialogue box select the mode of modifying the number of knots for a b-spline.

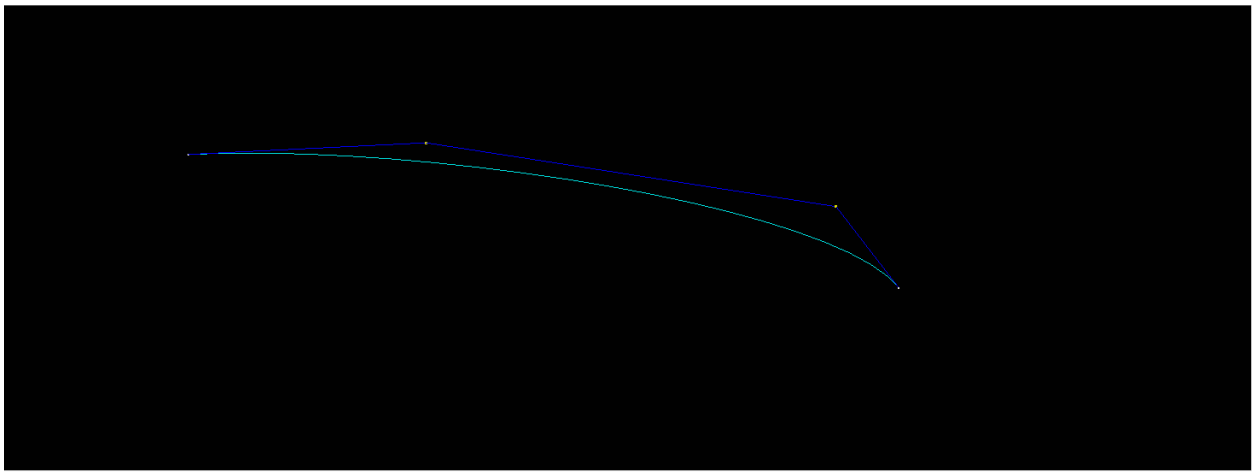



Keep shape exactly – enable or disable this option:

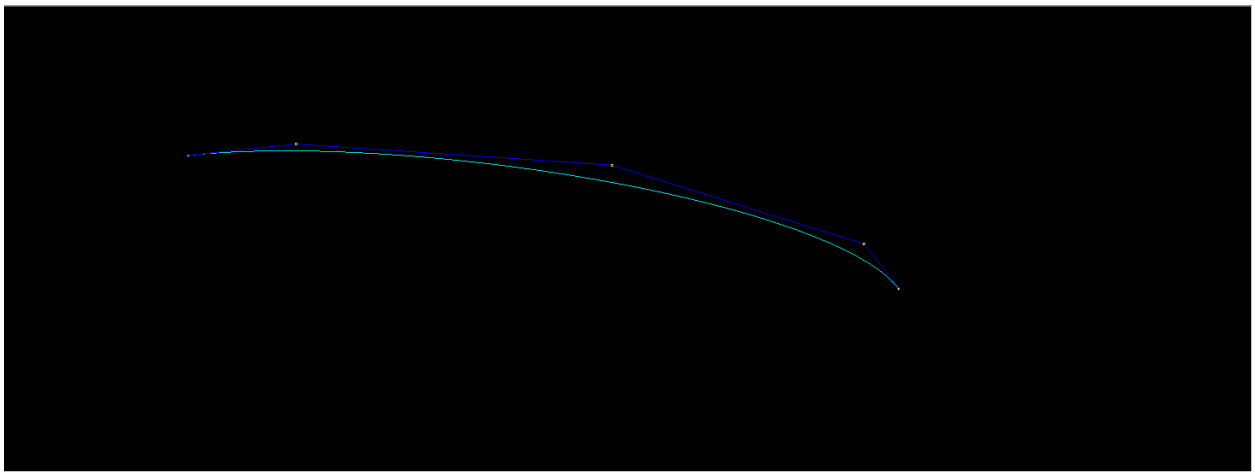
Knot to:

Add - add a knot on the line. Input position of the new knot.

Before:

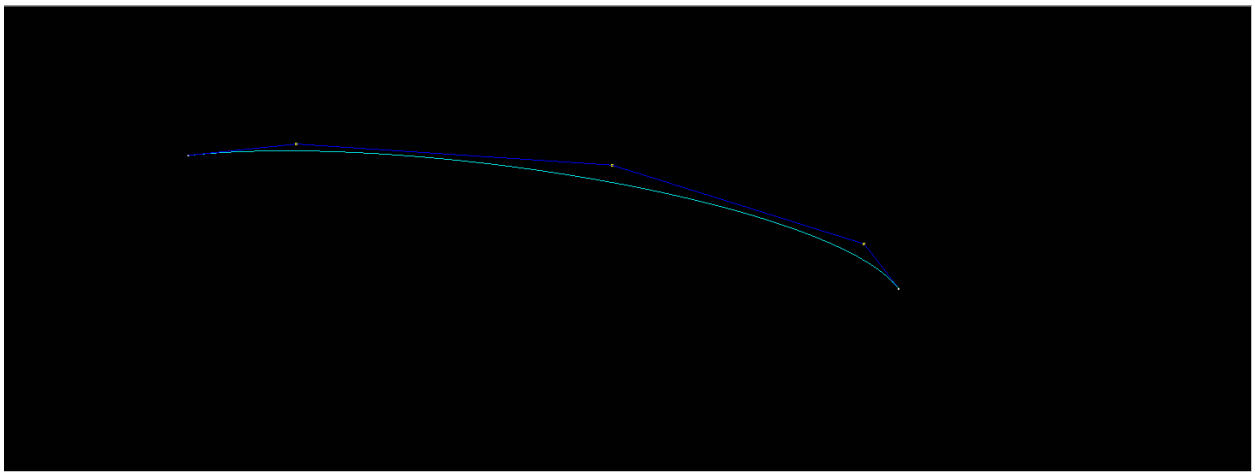


After:

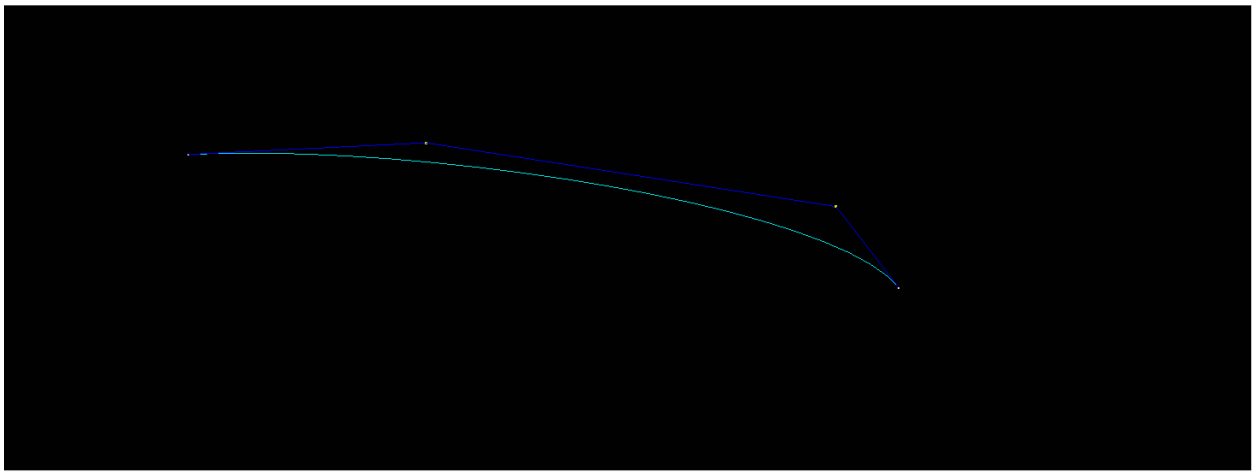


**Delete -** delete a knot from the line. Select knot to delete.

Before:



After:



On deleting the knot the shape of the line will be changed.

Make even parameterization – make equal parameter distribution along curve. Number of knots have to be set by user.

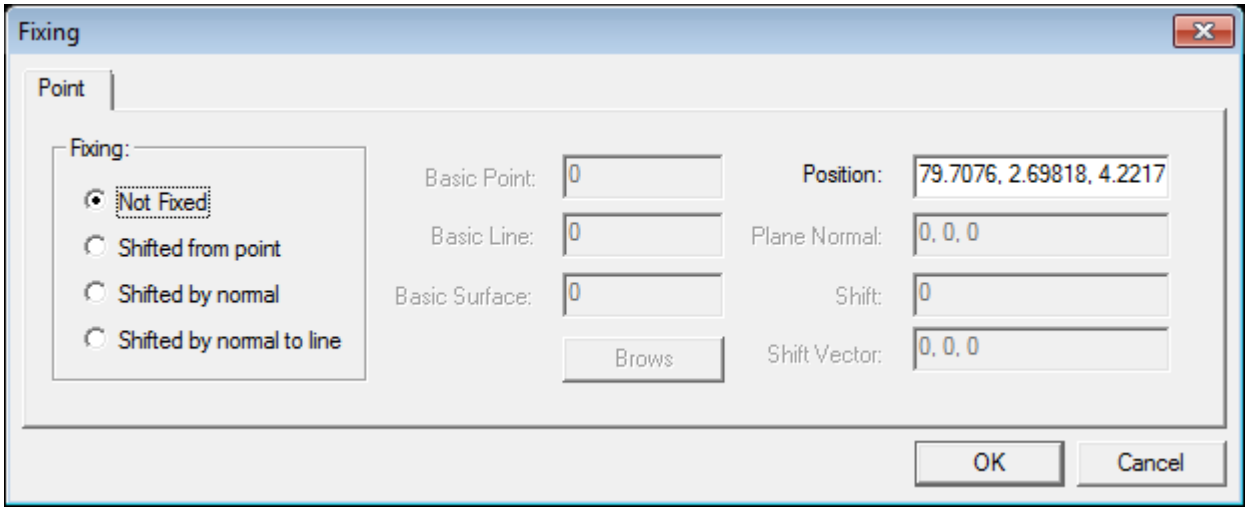
Two-stage method –advanced for better shape.

### Element ▸ Fixing

This command will assign the required fixation (subtype) to the corrected point or line and change parameters of fixation or parameters of the size.

Select an element to change the parameters or type of fixation.

Select the desired options and parameters for the spatial point in the dialogue box




Not Fixed – without fixation:

Position - position of the point.

Shifted from point – fixed on shift:

Basic point - 0 – internal point name (number);

Shift vector - 0, 0, 0 - vector of shift from the basic point.

Shifted by normal – fixed on shift by normal to surface:

Basic point - 0 – internal point name (number);

Basic surface - 0 – internal surface name (number);

Shift - 0 – distance from the basic point.

Shifted by normal to line - fixed on shift by normal to the line projection:

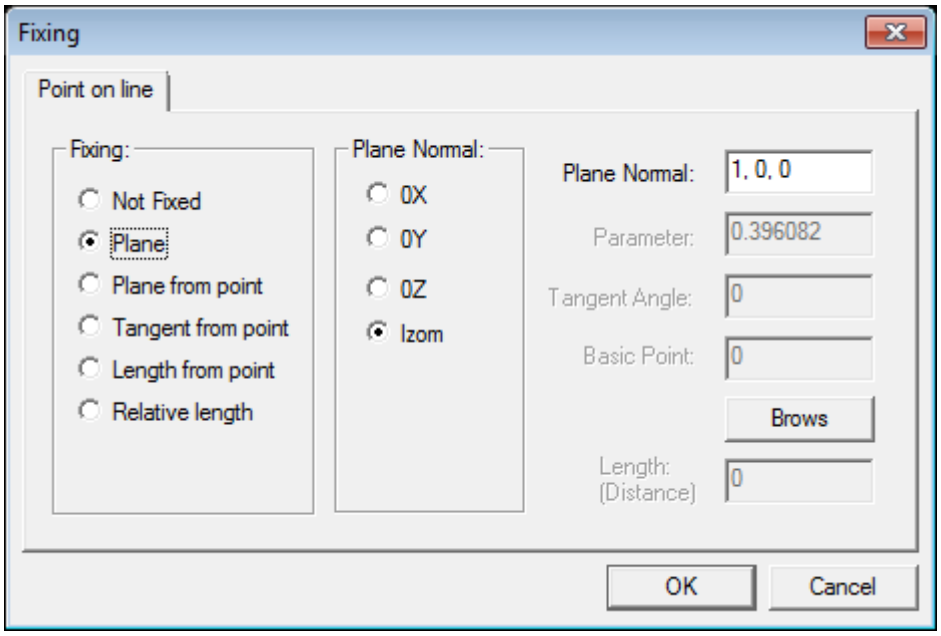
Basic point - 0 – internal point name (number);

Basic line - 0 – internal line name (number);

Plane normal - 1, 0, 0 – vector of normal to determining plane;

Shift - 0 - distance from the basic point.

Select the desired options and parameters for the hanging point in the dialogue box




Not Fixed - without fixation:

Parameter - 0 – parameter of the point on the reference line.

Plane – fixed by coordinate (in plane):

Plane normal - 1, 0, 0 – vector of normal to plane.

Plane from point – fixed on shift by one coordinate (in plane with shift):

Plane normal **-** 0, 0, 0 - vector of normal to plane;

Basic point - 0 – internal point name (number);

Distance - 0 - distance from the basic point to the plane.

Tangent from point – fixed on tangent to reference line:

Plane normal - 0, 0, 0 - vector of normal to plane;

Tangent angle - 0 – angle of inclination to tangent;

Basic point - 0 – internal point name (number);

Length from point – fixed by length:

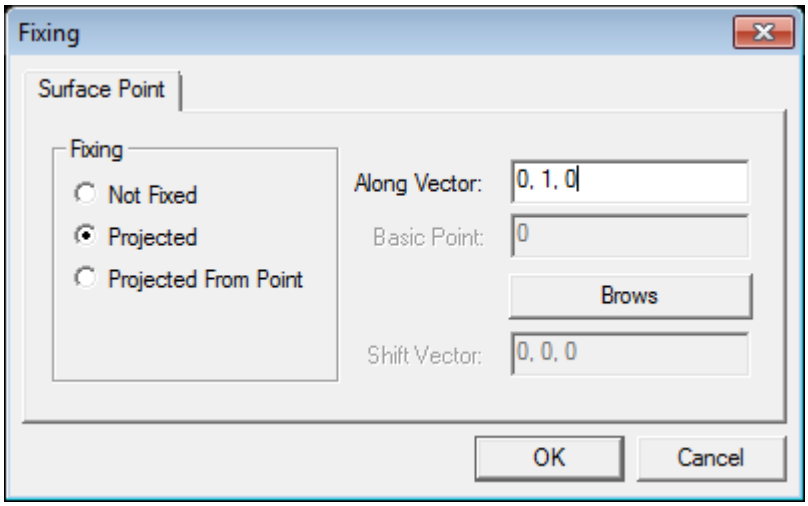
Basic point - 0 – internal point name (number);

Length - 0 – fixed length.

Relative length – fixed by relative length of the line:

Length - 0 – fixed relative length; value 0 corresponds to the beginning of the line and value 1 corresponds to the end of the line.

Select the desired options and parameters for the surface point in the dialogue menu:




Not Fixed - without fixation:

Projected – fixed by two coordinates (by vector):

Along vector - 0, 0, 0 – projecting vector.

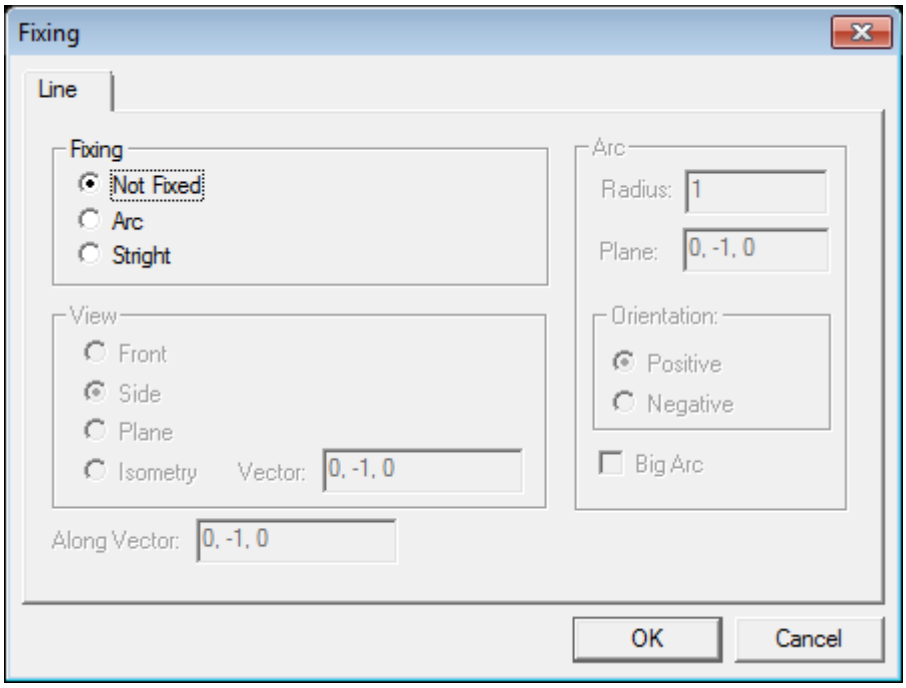
Projected From Point – fixed on shift by two coordinates (by vector with shift:

Along vector - 0, 0, 0 - projecting vector;

Basic point - 0 – internal point name (number);

Shift vector - 0, 0, 0 – shift vector of the basic point.

Select the desired options and parameters to change fixation of the line in the dialogue menu:




Fixing:

Not Fixed – without fixation;

Arc – obtaining and fixing of an arc shape;

Straight - obtaining and fixing of a straight line shape;

Vector – modifying the projecting vector (for surface lines).

Plane - 0, 1, 0 – vector of normal to plane on which the projection of line has the shape of an arc or a straight line;

Radius (m) - 0 – arc radius;

Orientation - positive/negative - orientation of the arc convexity;

Big arc - no – selecting an arc with a smaller or larger central angle:

Positive Negative Big arc

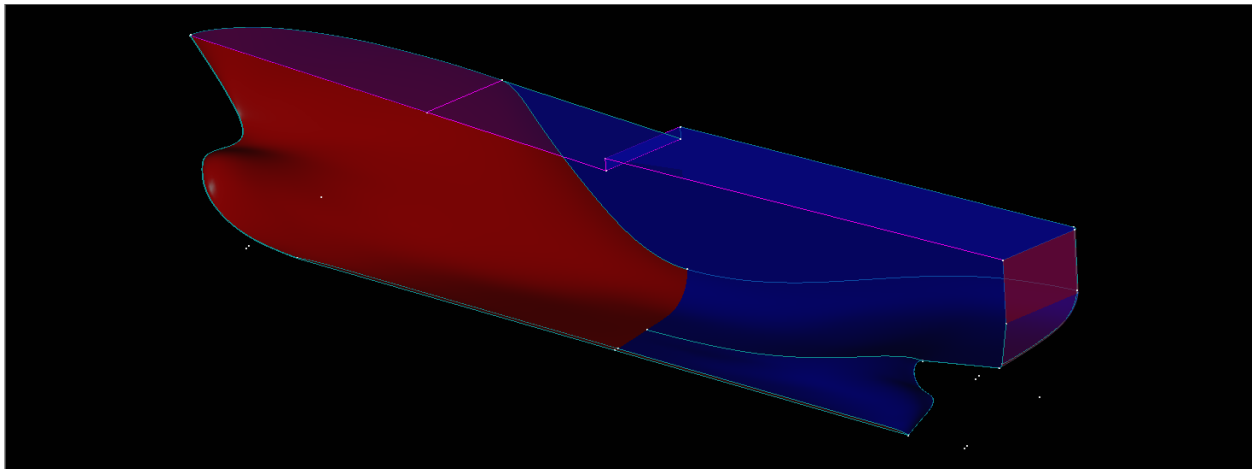
**Along vector -** 0, 1, 0 – projecting vector (for surface lines).

### Element ▸ Surface Orientation

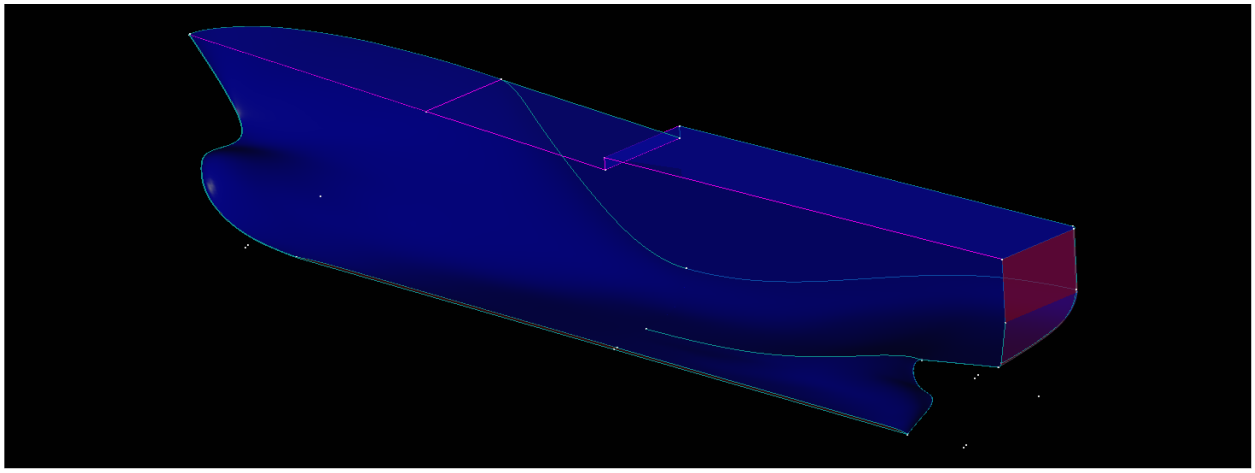
This command allows changing surface orientation.

Choose a surface which necessary to change orientation. To stop the process, press Esc or right-click the mouse.

Before:



After:

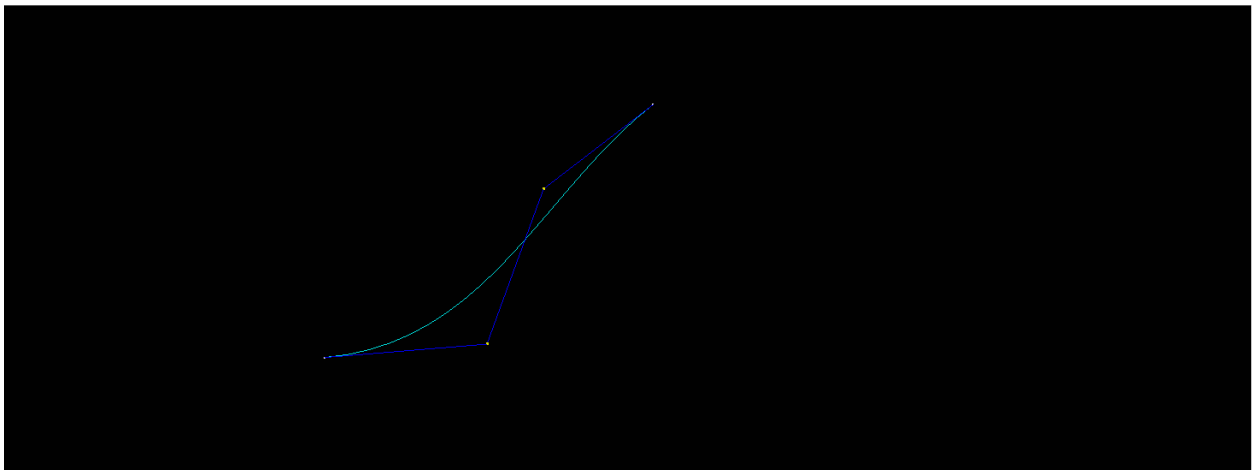


### Element ▸ Ortho Line

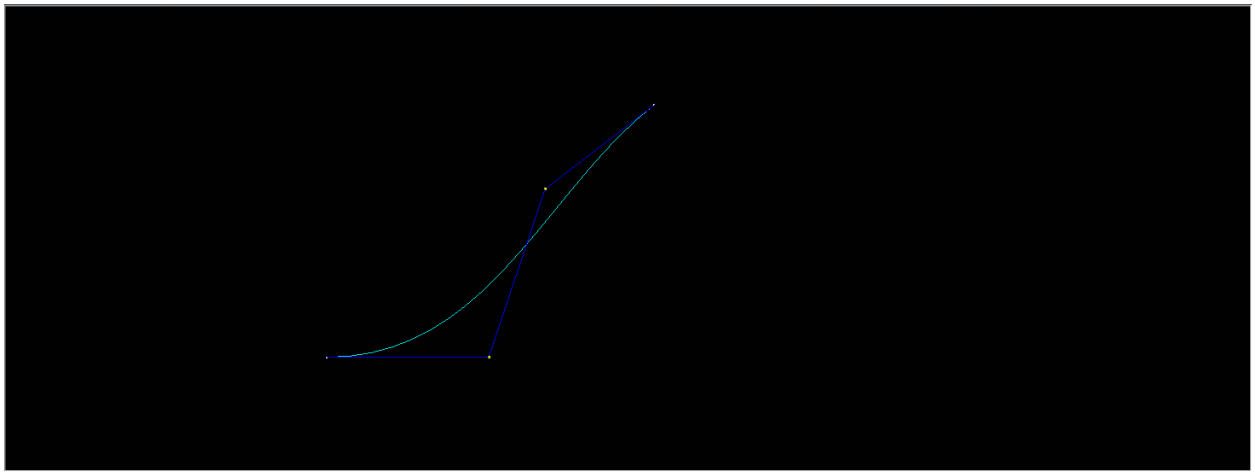
This command sets tangents to line in the end points vertically or horizontally in the working plane.

Select a line to be edited. Select the first knot closest to end point of the line. Tangency in this point will be set up vertically or horizontally depending from previous position of the control point".

Before:



After:

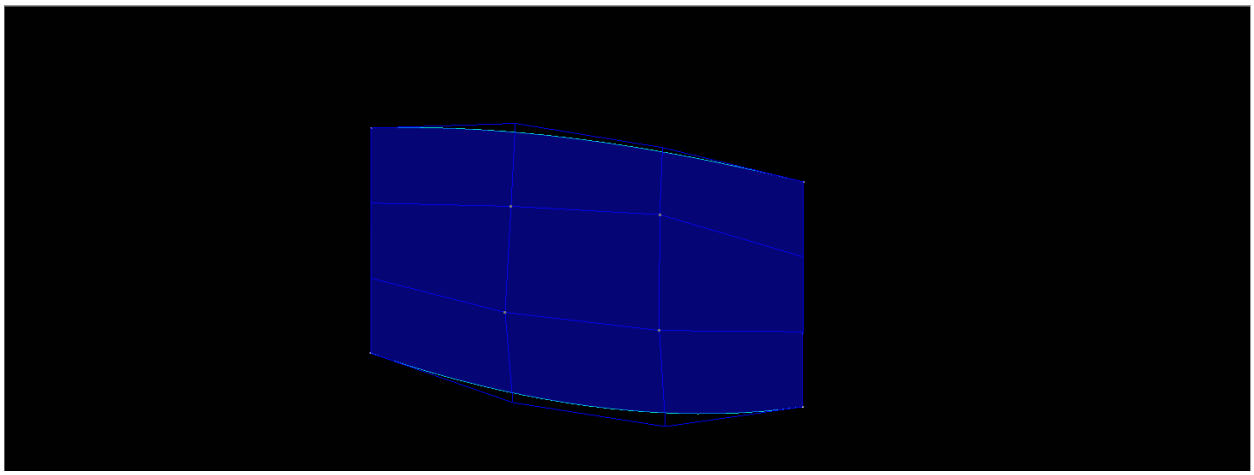


**Element ▸ Ortho Surface**

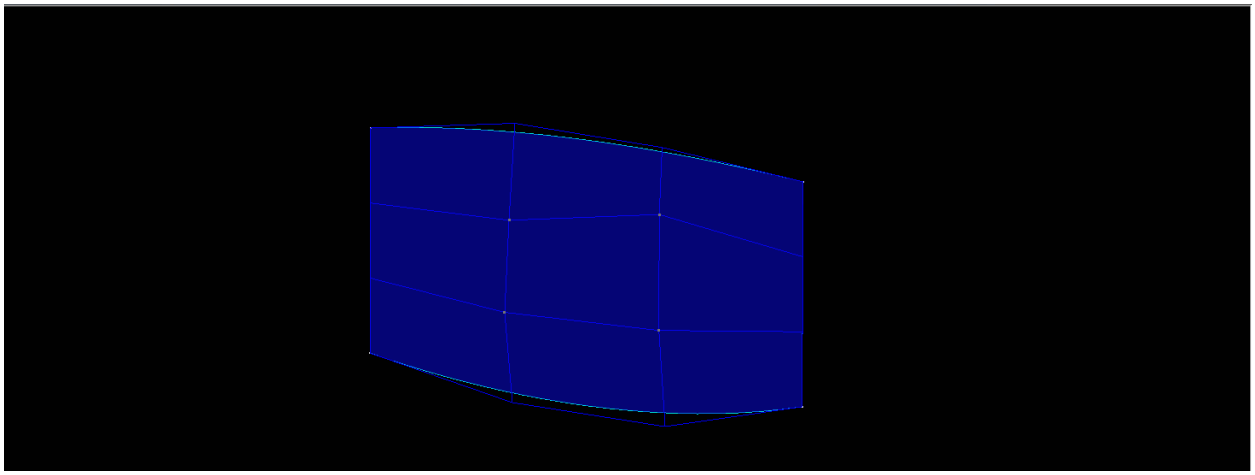
This command sets an auxiliary mode of editing the surfaces for boundary knots.  After modification such knot will be “vertical” or “horizontal” in the working plane.

The auxiliary surface editing mode will be enabled. Select a surface to be edited. Select a knot to be set in the orthogonal position.

Before:



After:



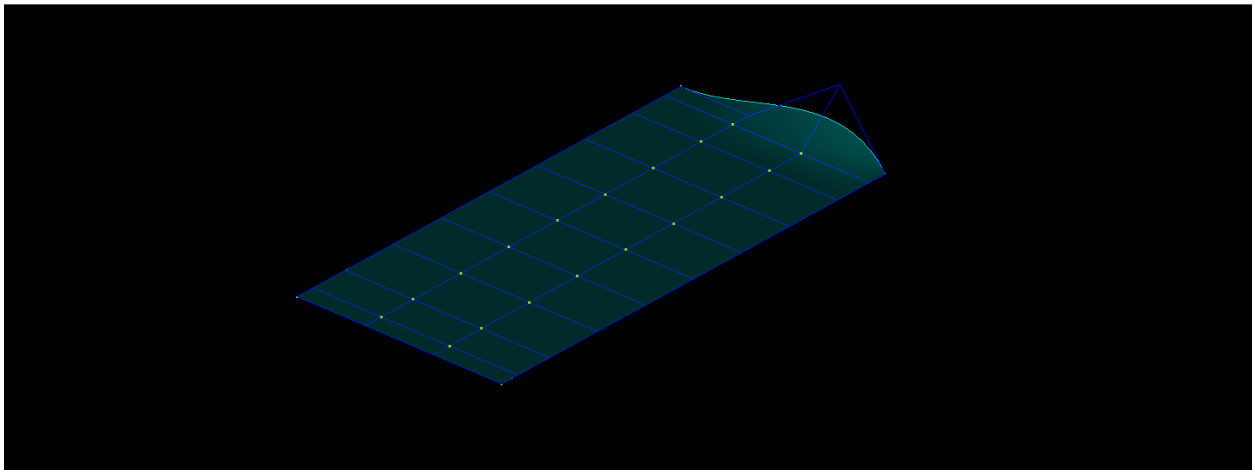
To switch off  the mode, click the Ortho Surface command again.

### Element ▸ Tangent

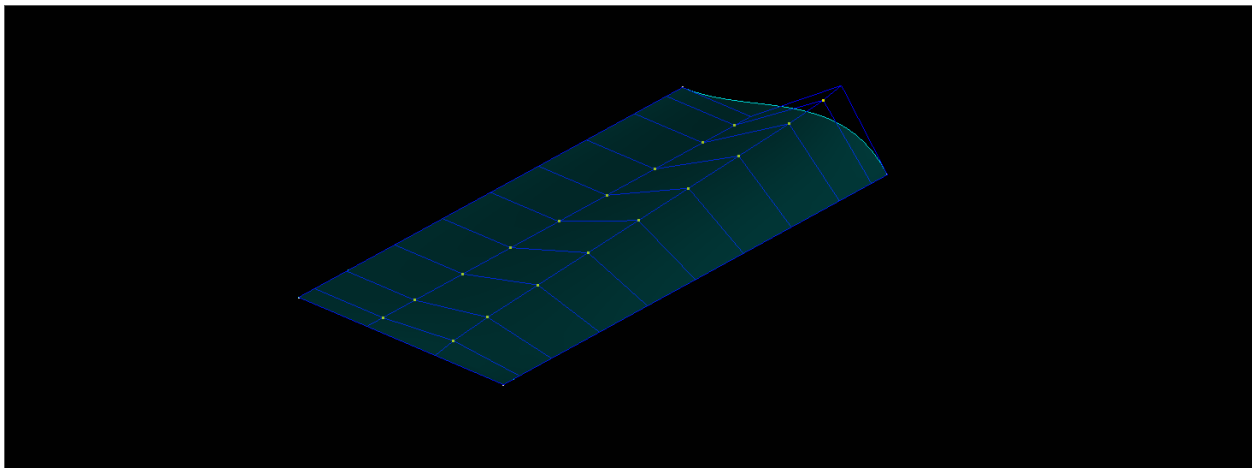
This command is use to set linear depends for tangency along surface's boundary.

Select begin point of the boundary knot. Select end point of the boundary knot.

Before:



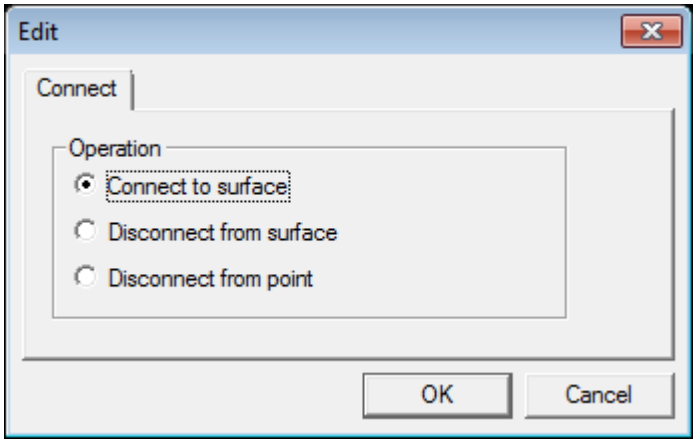
After:



**Element ▸ Connect**

This command makes the topological of the elements.

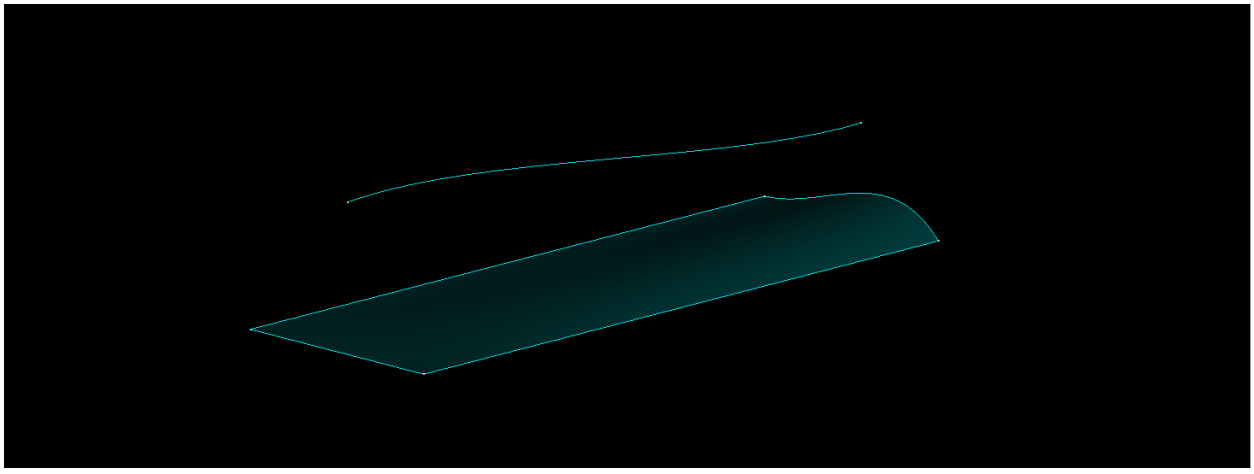
Three methods of transformation are available:



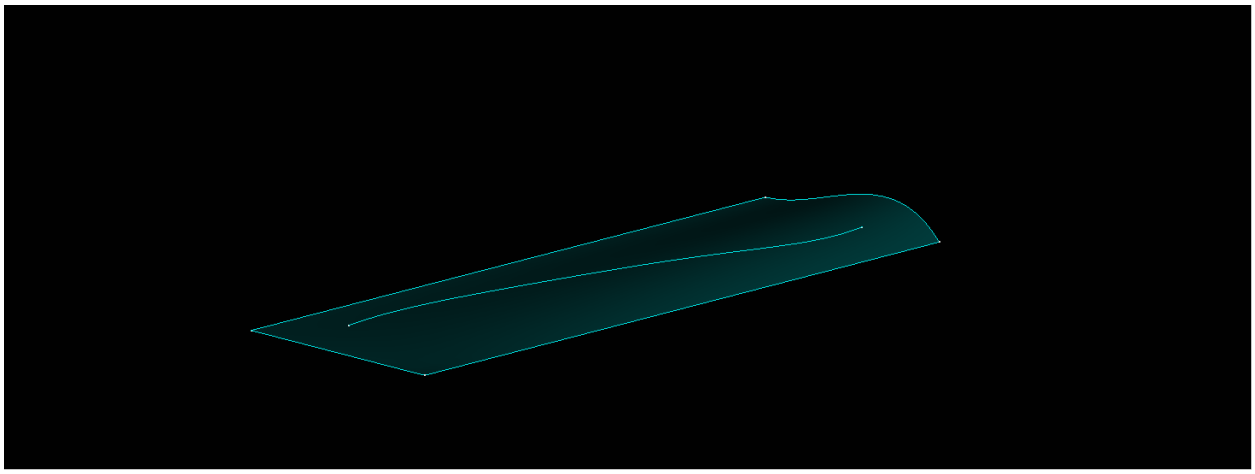

Connect to surface:

This option transforms spatial points and lines into surface points and lines. The points and lines are projected on the surface along the vector perpendicular to the current working plane.

Before:



After:



Disconnect from surface:

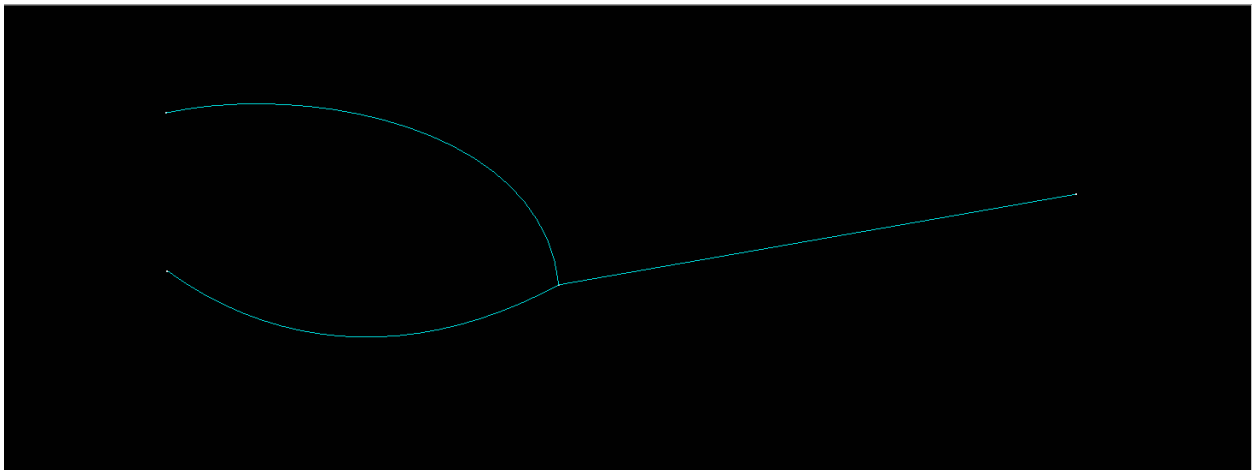
This option transforms surface points and lines and hanging points into spatial points and lines.

Disconnect from point:

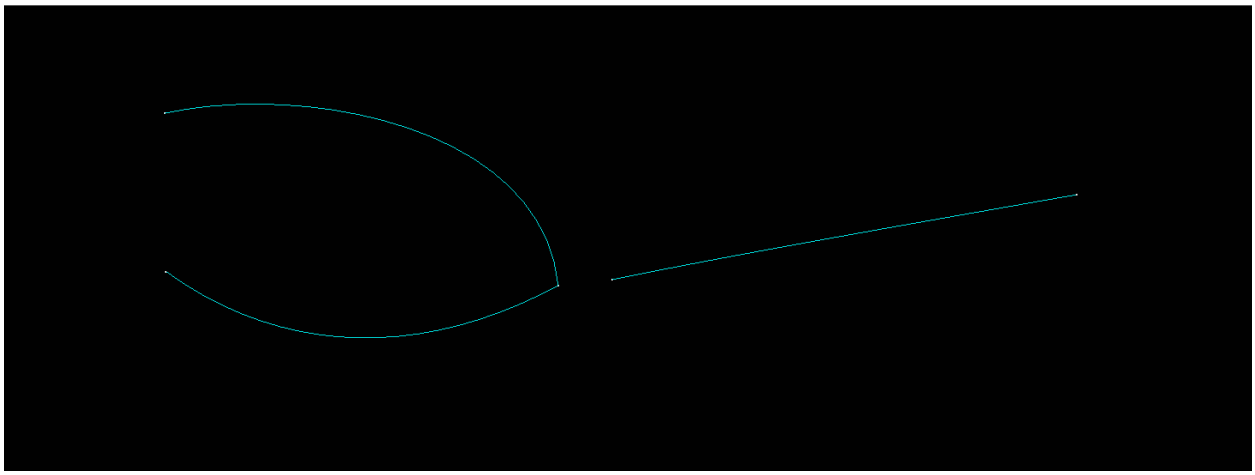
This option “divides” the point into two points thus making the topological dependency of a certain line on other elements unnecessary.

Select point. Select disconnecting line.

Before:



After:

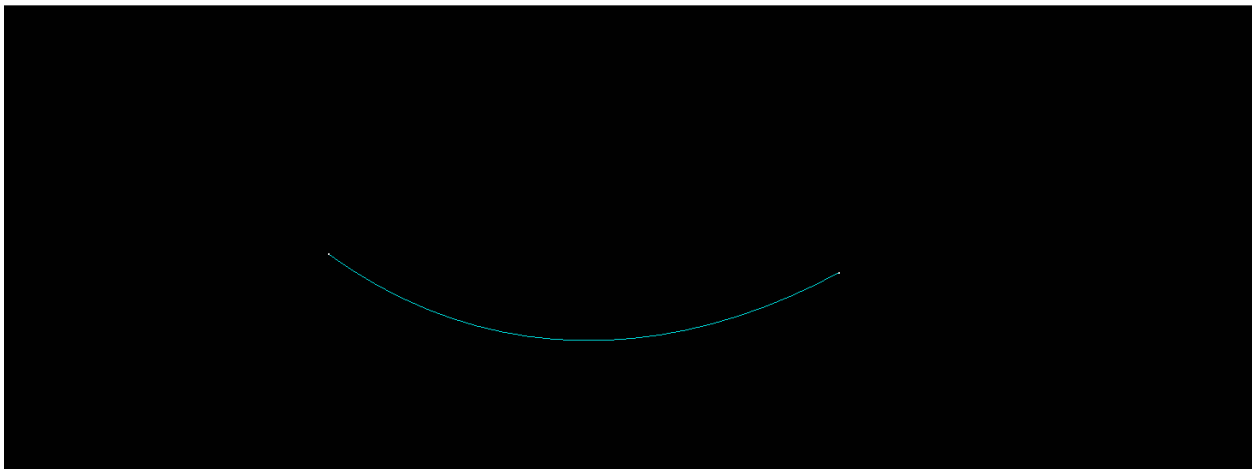


**Element ▸ Extend**

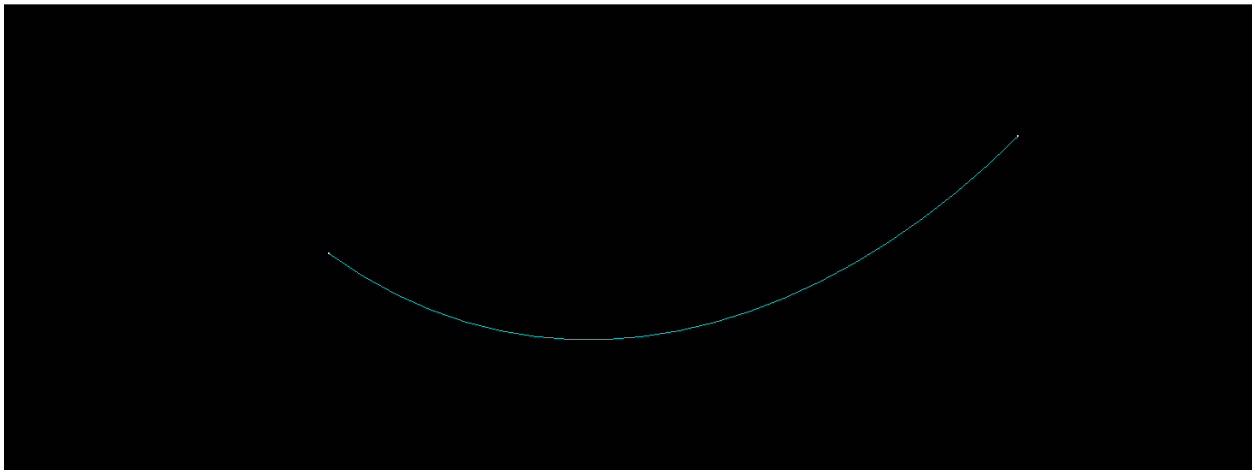
This command extends or shortens selected line without changing the shape.

Select a line. Select line’s end point. Move the cursor to extend or shorten the line.

Before:



After:



**Element ▸ Knuckle**

This command is used for making knuckles inside surfaces and lines.

For line:

Select line for edit. Select control point where is necessary to have a knuckle.

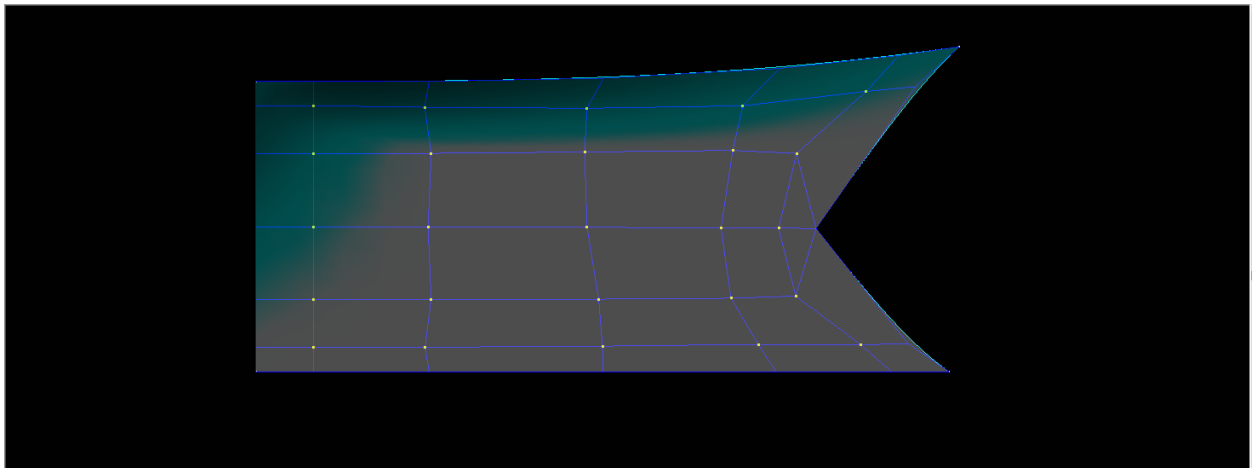
Before:

After:

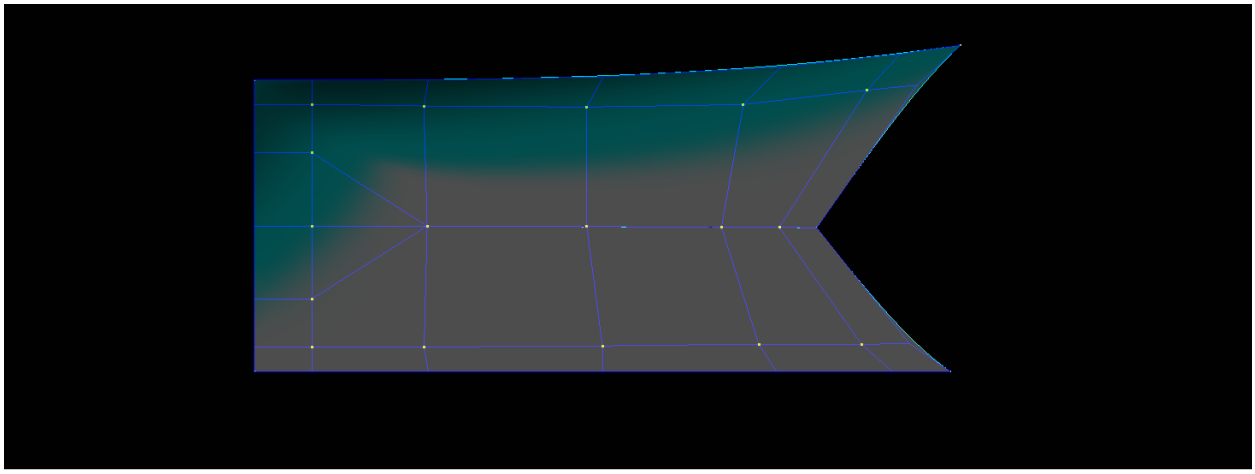
For surface:

Select surface for edit. Select begi and end points of the row to make knuckle.

Before:



After:



### Element ▸ Copy Geometry

This command copies geometrical data from one source element to another.

Select a source element. Select modified elements.

### Element ▸ Update

This command is used for modification of the connected elements without finishing with editing of the master element.

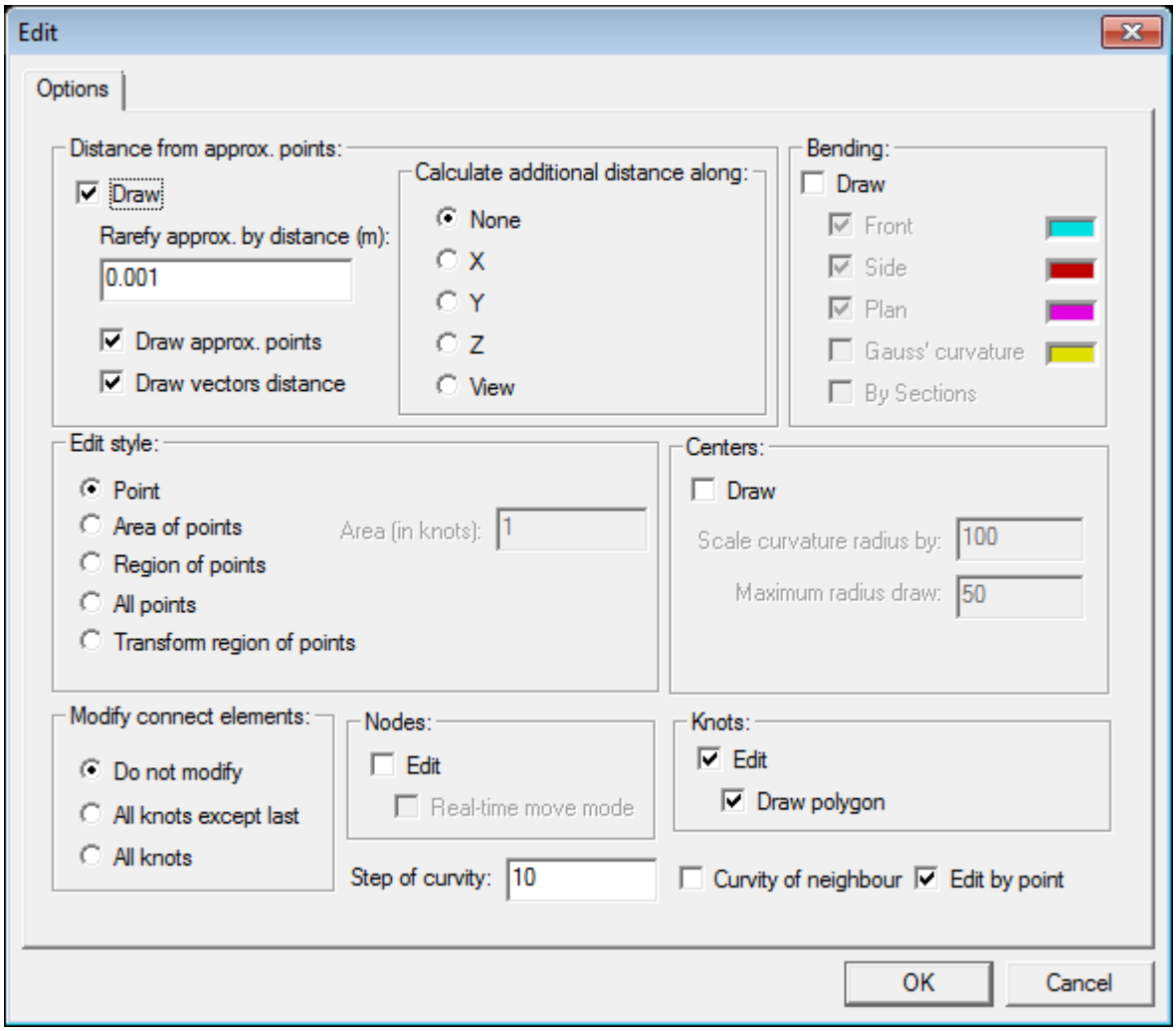
### Element ▸ Rebuild

This command is used for redraw surface during editing.

**Element ▸ Options**

This menu contains various methods and options for manual correction of lines and surfaces.

Select parameters in the dialogue box.




Distance from approx. points: - visualization and calculation deviations from approximation points.

Draw – visualization points and deviations

Rarefy approx. by distance (m) - 0.001 – the value which determines the minimal distance between the used approximation points.

Draw approx. points - draw approximation points

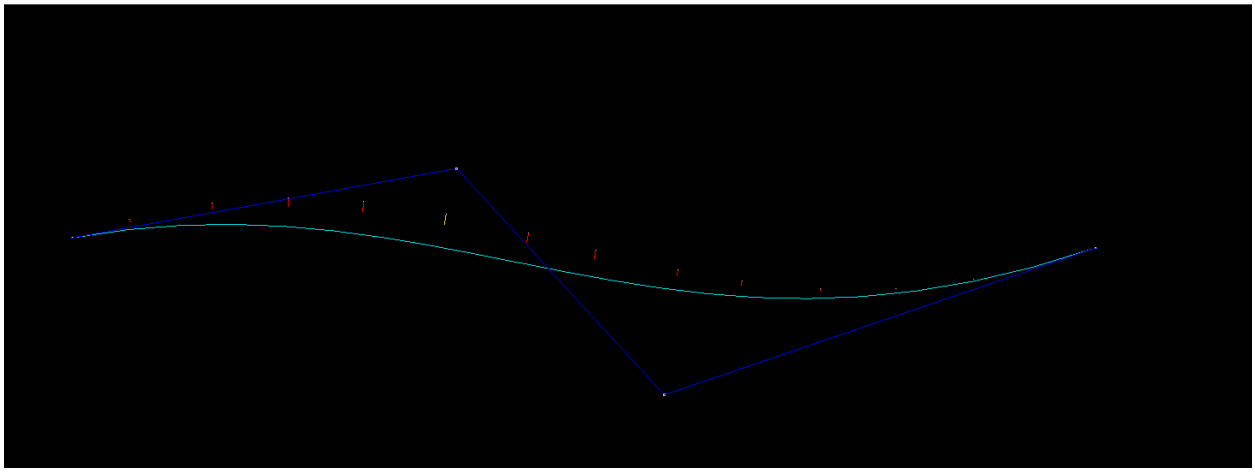
Draw vectors distance - draw deviations vectors

Calculate additional distances along:

None – makes it possible during the correction of surfaces, in addition to the distance in 3D space.

X, Y, Z – along x, y and z axes respectively;

View – along the sight direction (e.g., in isometry).



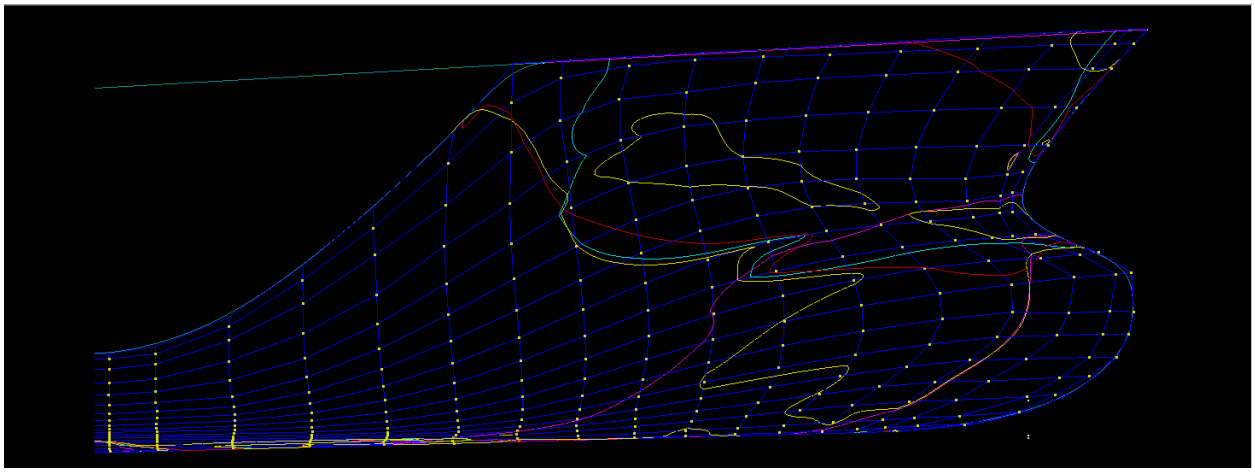
Bending:

Draw - enable/disable inflection lines visualization

Front**,** Side**,** Plan - visualization according selected projection

Like Sections - visualization according sections

Gauss’ curvature - visualization according Gauss’s curvature



Edit style:

Single point – single control point modification.

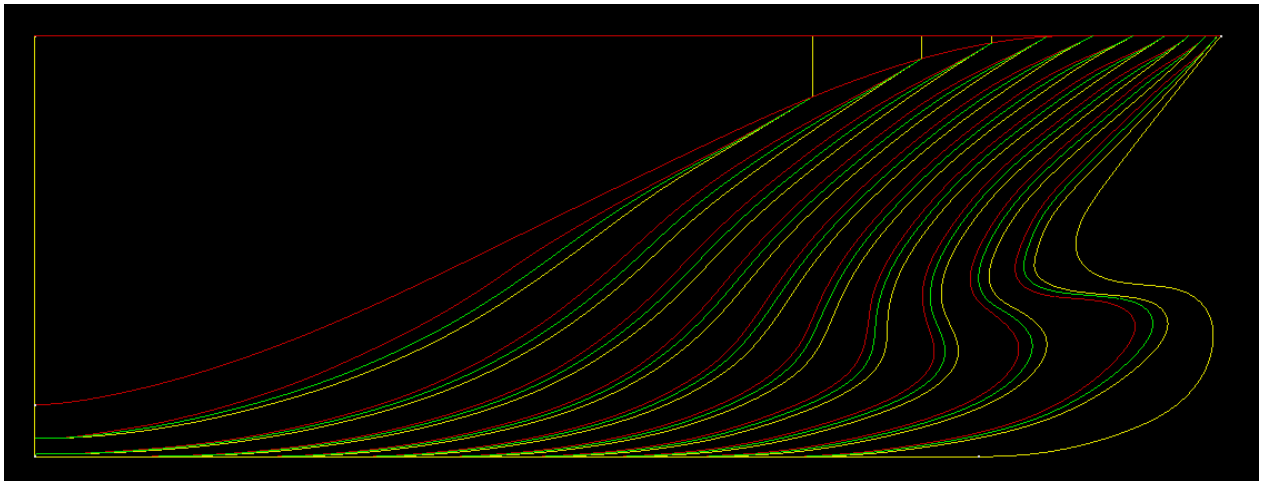
Area of points – area of the control points modification.

Area (in knots): 2 – number affected Bezier patches. The farther is the knot from the edited knot, the shorter is the distance that it moves. Boundary knots do not move to keep tangents at the borders of lines unchanged.

Region of points – similar to the previous mode movement of one knot affects a few adjacent knots.

All points – the same as the previous mode, but editing of one knot affects the whole surface.

Transform region of points – this mode moves all the knots within the user specified area to an direction vector and provide adding or reducing of volume.

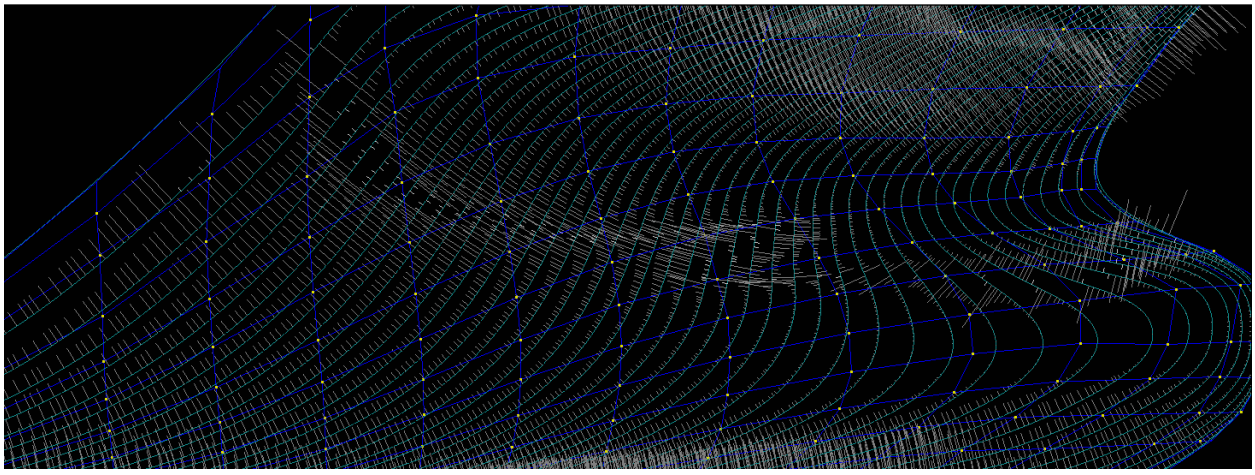


Center:

Draw - Display the curvature and line bend points (in projection to the working plane) or surface sections.

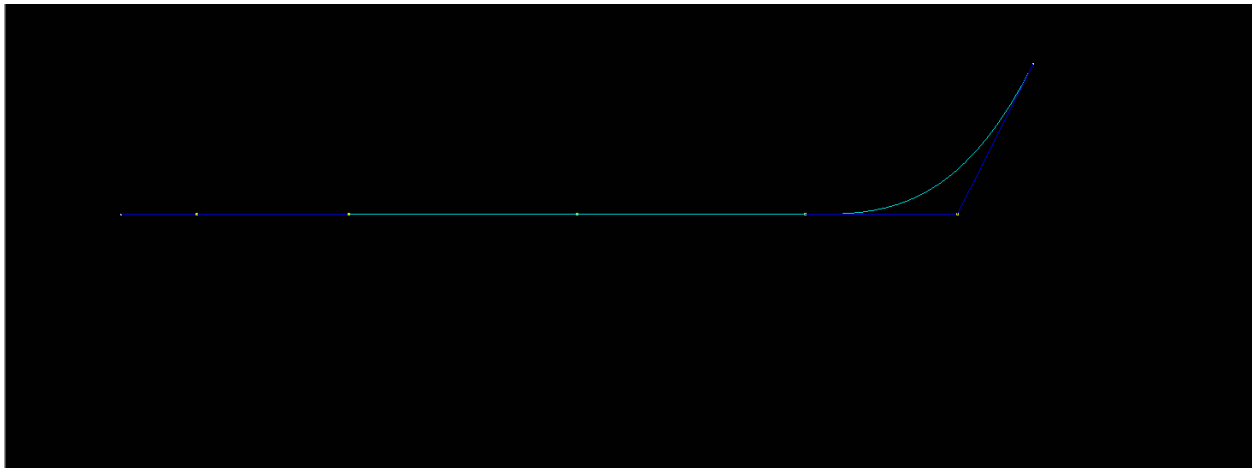
Scale curvature radius by - 100 – scale parameter for curvature. The value is the number of times the length of the segment representing the curvature in the point is less than the radius of curvature in the given point.

Maximum radius draw - 50 – the maximal radius of curvature displayed on the screen.

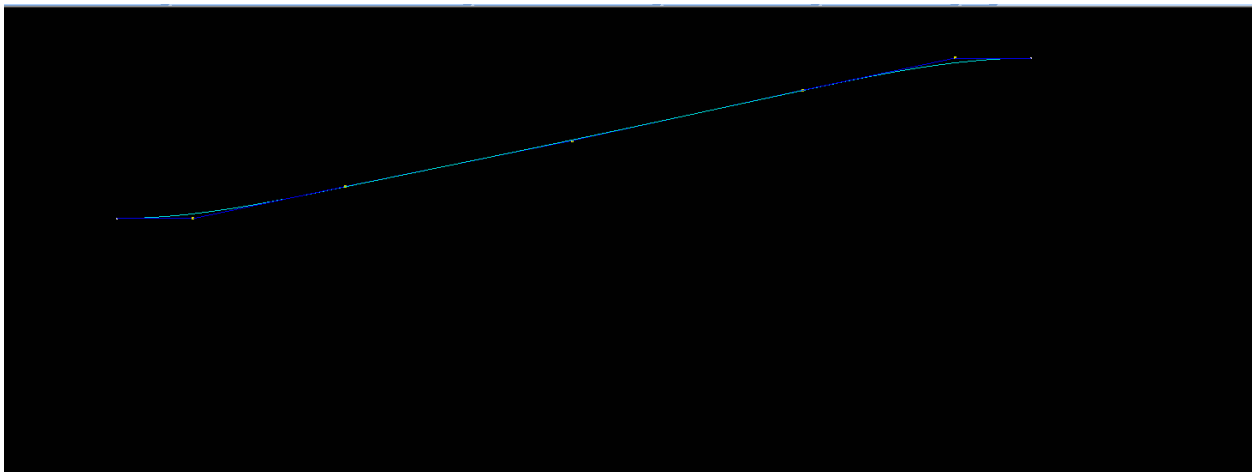


Modify connect elements: this option set up medication rules for linked elements. For example, after changing point position, shape of all lines (connected to the point) will be modified according following rules:

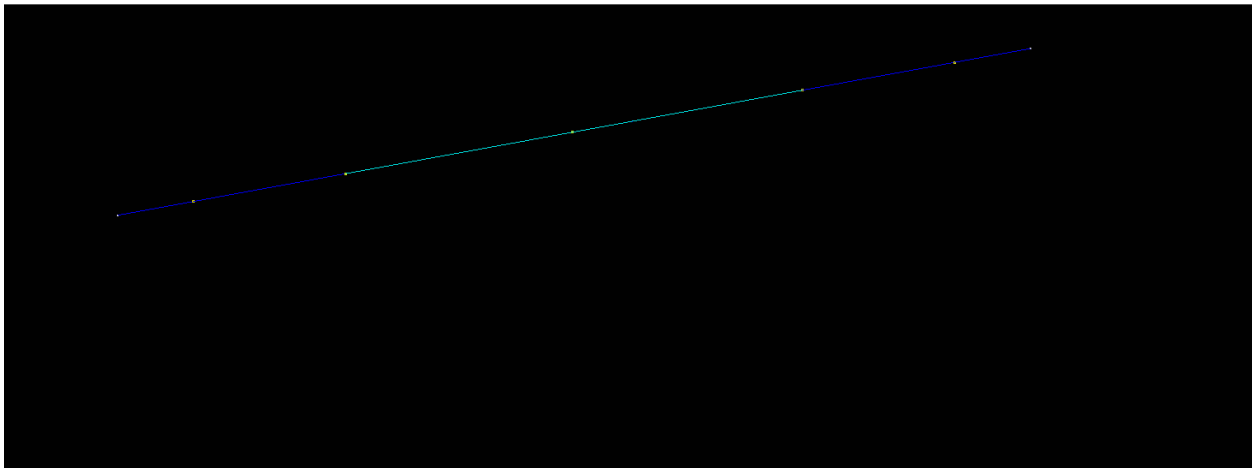
Do not modify – means only end point of the line will be modified. All other control points will keep same positions.



All knots except last – means only end point of the line will be modified.



All knots – all knots will be modified by linear law.



Contents of the given dialogue box can conventionally be subdivided into the following sections: correction by knots, nodes.

Nodes:

Edit - allow editing by nodes.

Real-time move node:  this parameter indicates whether the whole polygon of the surface will be redrawn while moving the marker during the correction of surface by B-spline node (this may be quite slow);

Knots:

Edit - allow editing by knots.

Draw polygon - allow knots visualization as polygon.

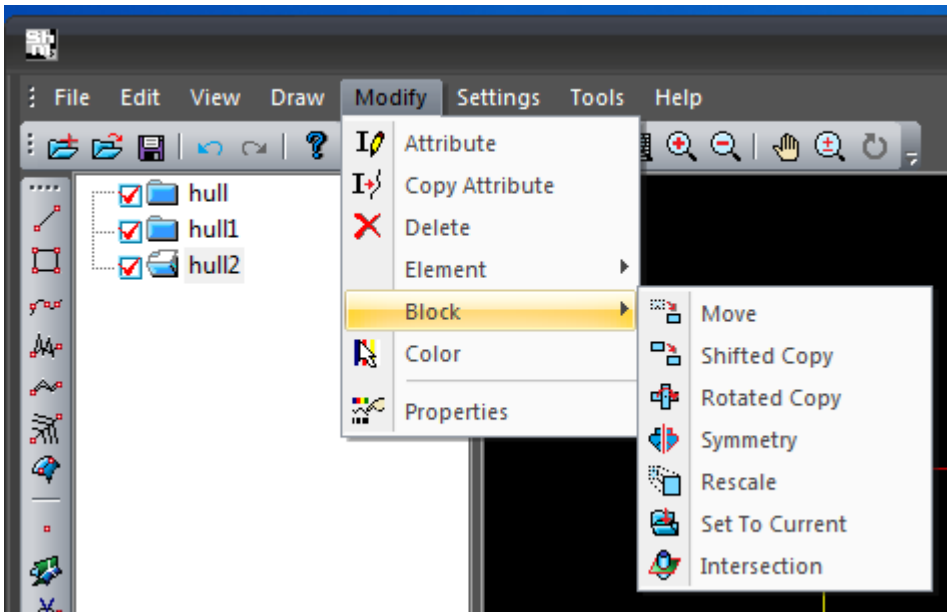
Step of curvature – 10 – number of steps for each curve (surface) segment for curvature calculation.

Curvature of neighbors – allow neighbors curvature elements visualization.

Edit by point: correction by random point.

## Modify ▸ Block

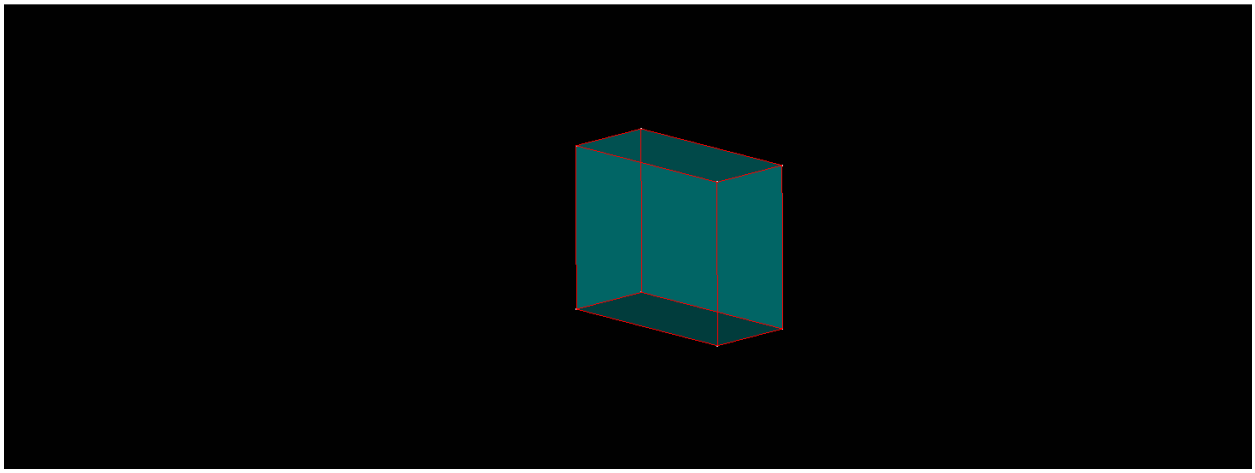
Menu Block contains the commands for various operations on a group of elements



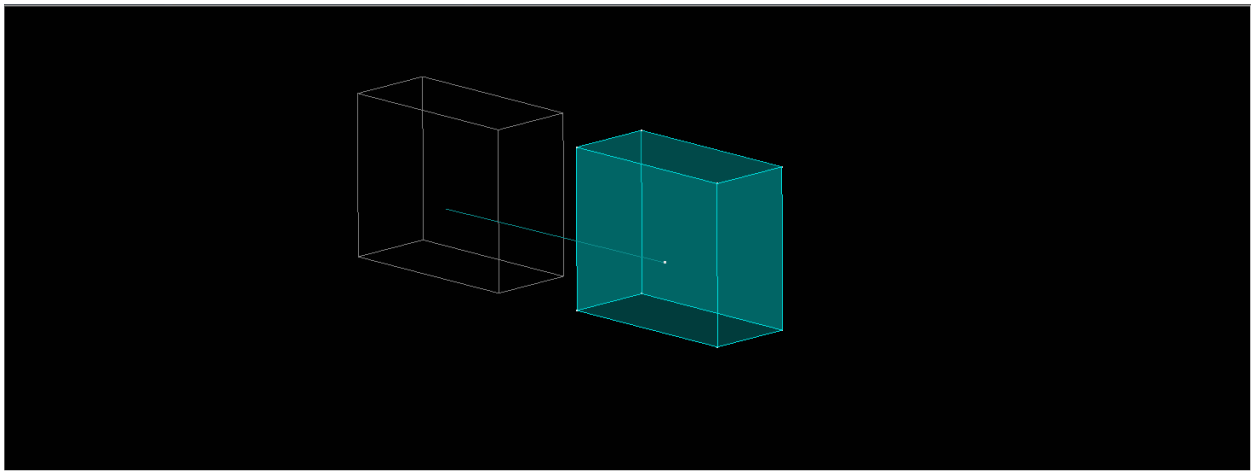

### Block ▸ Move

This command moves and rotates groups of elements in the working plane.

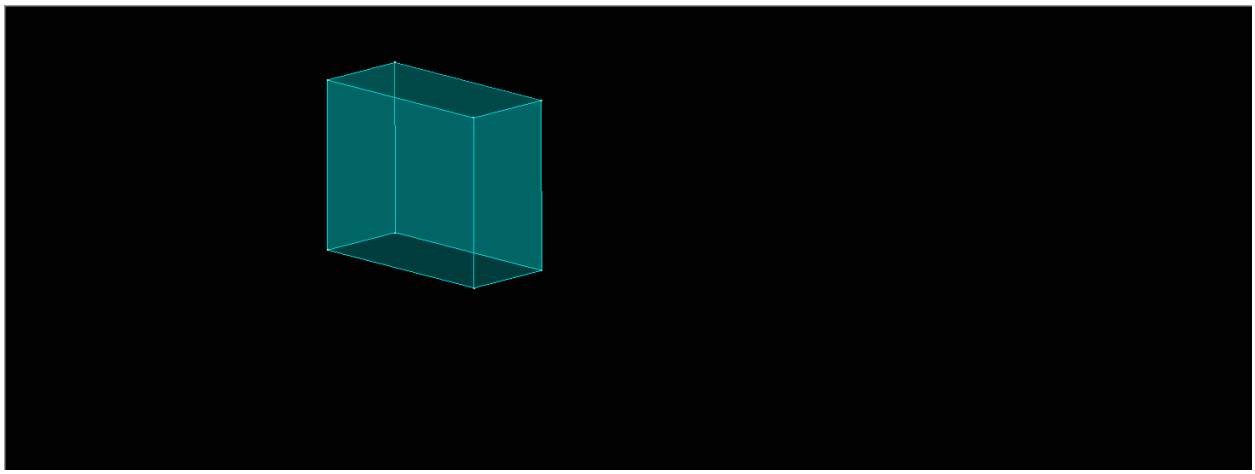
Select the elements to be moved. Press Enter after finishing selection.



Select basic point and move object to new position.

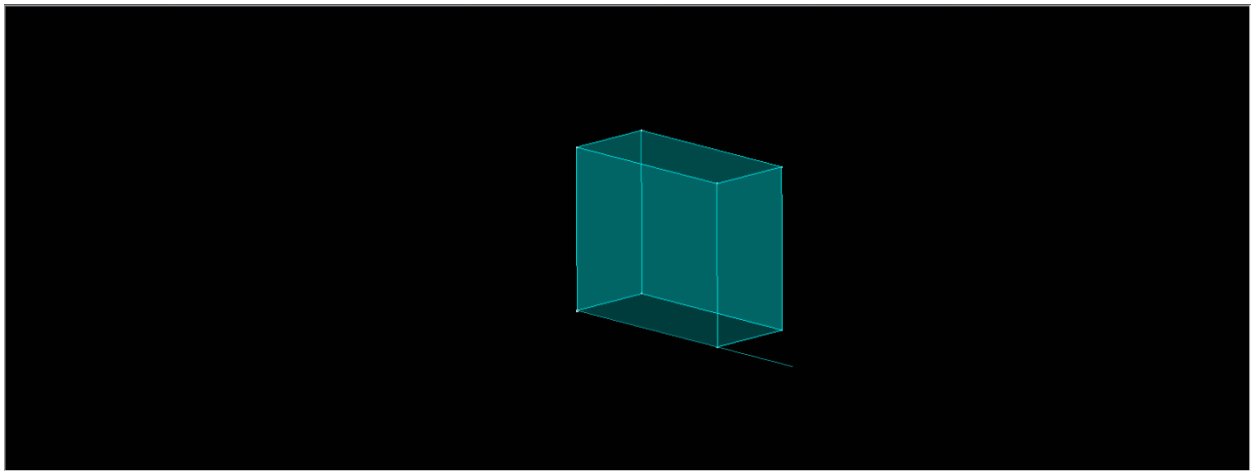


Select a new point.

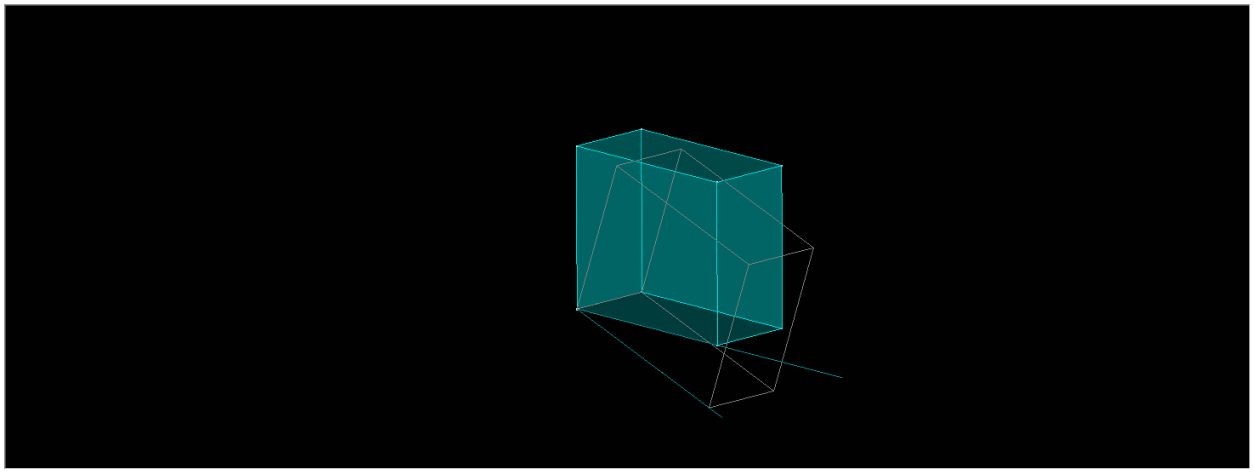


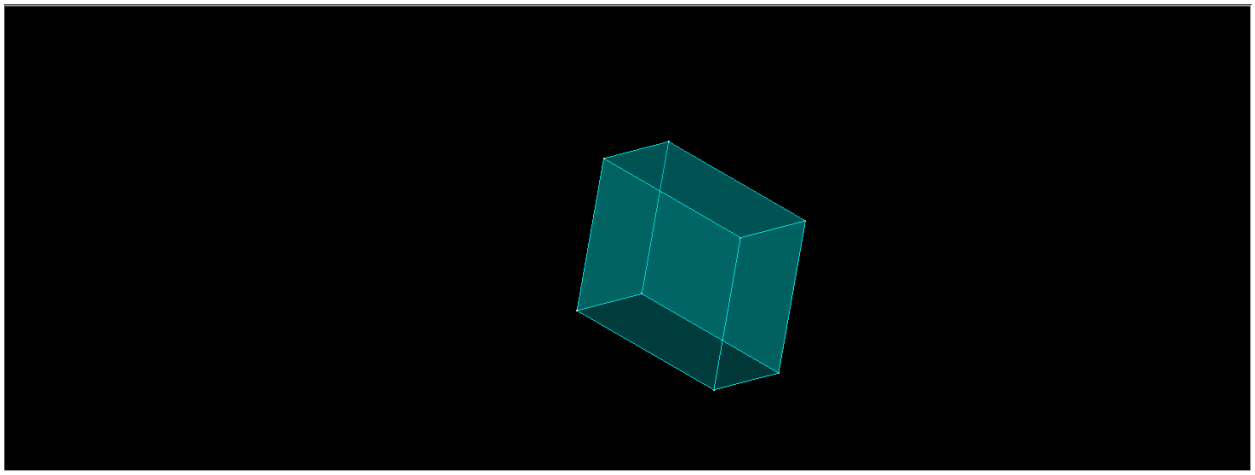
The elements image in a new position will appear in the model.

If required, move to rotation press Tab key. Specify the center of rotation. Input a base angle of rotation.



Specify the angle or rotation.



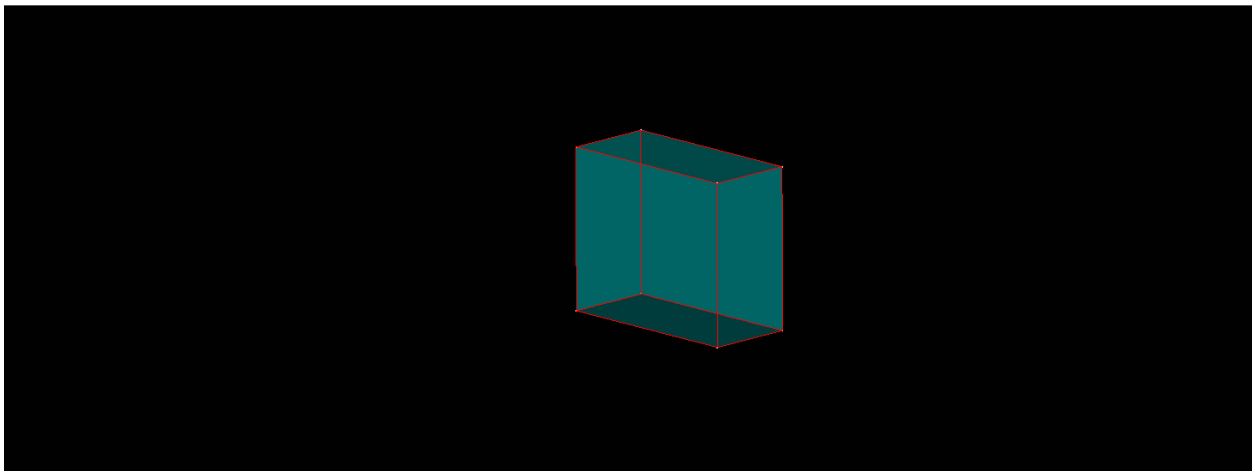


The elements image in a new position will appear in the model.

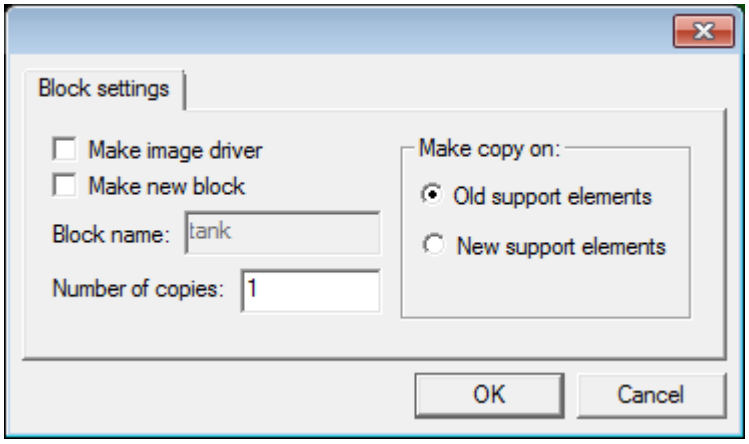
### Block ▸ Shifted Copy

This command makes shifted copy of the selected elements.

Select the elements to be copied. Press Enter after finishing selection.



Select copy options in the dialogue box.




Make image driver - creates an image driver.

Make new block - creates a block for the copied elements.

Block name - name of the new block.

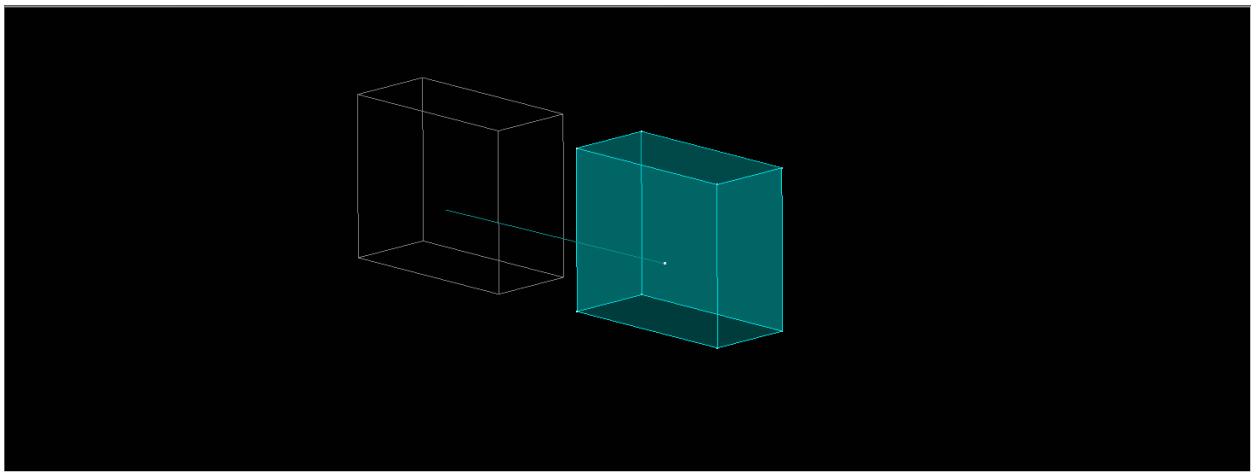
Number of copies **-** number of copies to be created. If the number is more than 1 the specified number of copies will be created. The distance between the copies will be the same as between the selected elements and the first copy.

Make copy on:

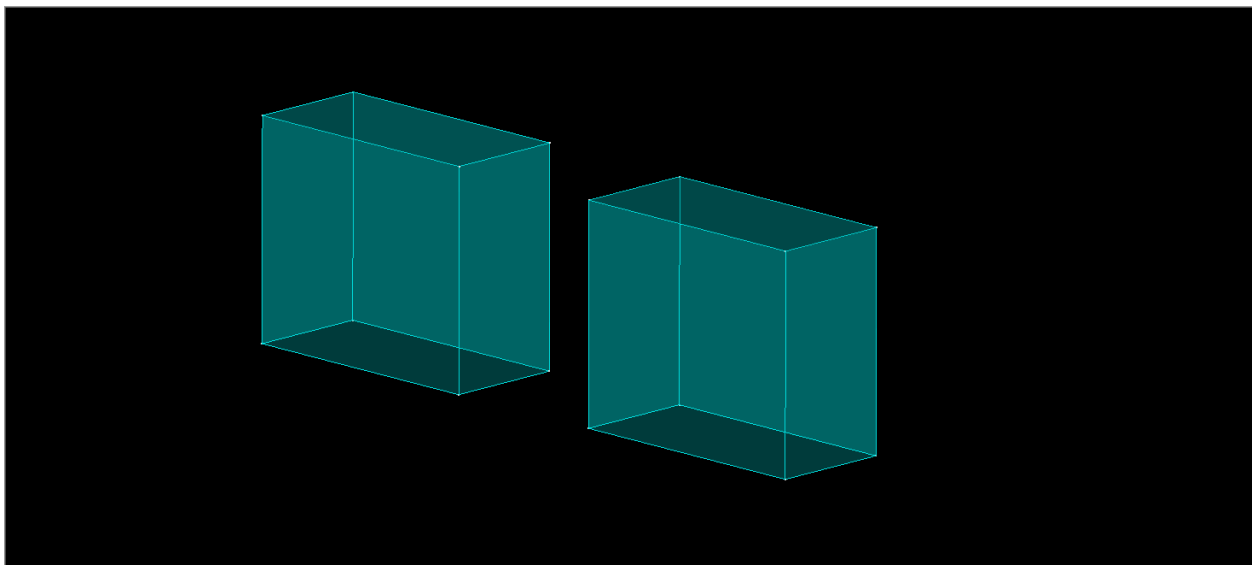
Old support elements – creates a copy with the use of the source reference elements.

New support elements - creates copies without the use of the source reference elements.

Select basic point and copy object to new position.



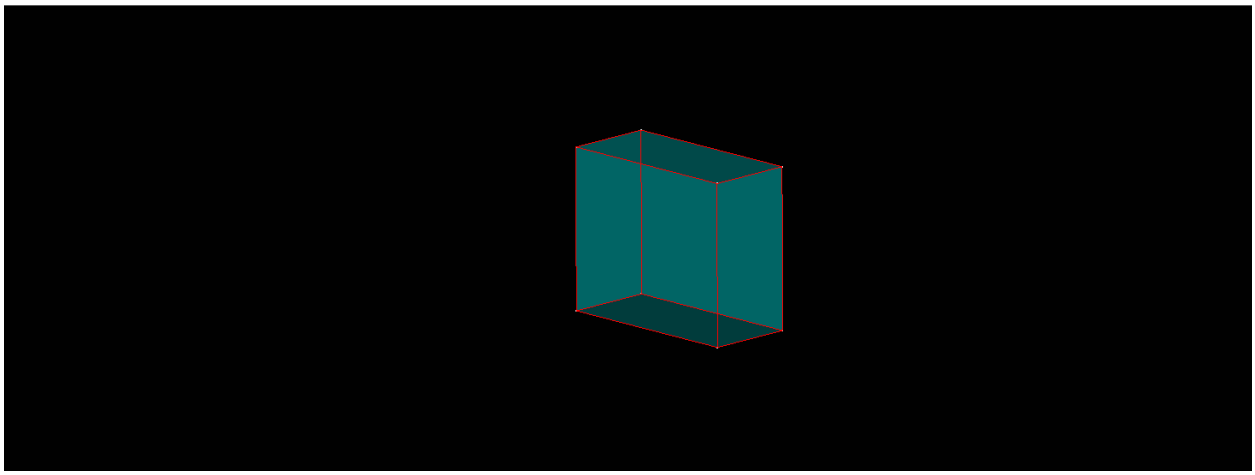
Copied elements will appear in the model.



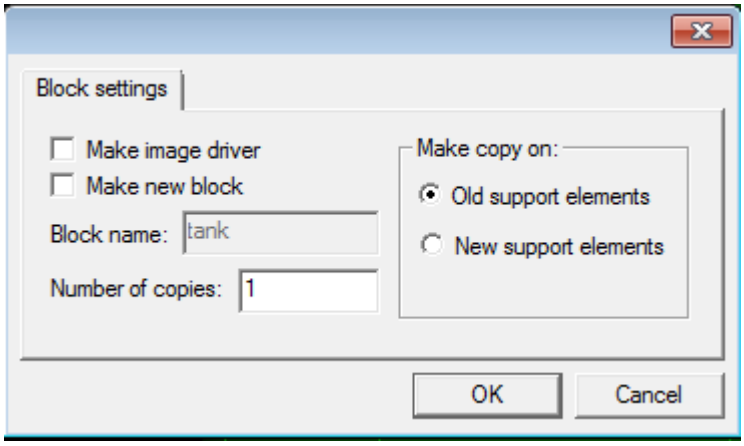
**Block ▸ Rotated Copy**

This command makes rotated copy of the selected elements.

Select the elements to be copied. Press Enter after finishing selection.



Select copy options in the dialogue box.




Make image driver - creates an image driver.

Make new block - creates a block for the copied elements.

Block name - name of the new block.

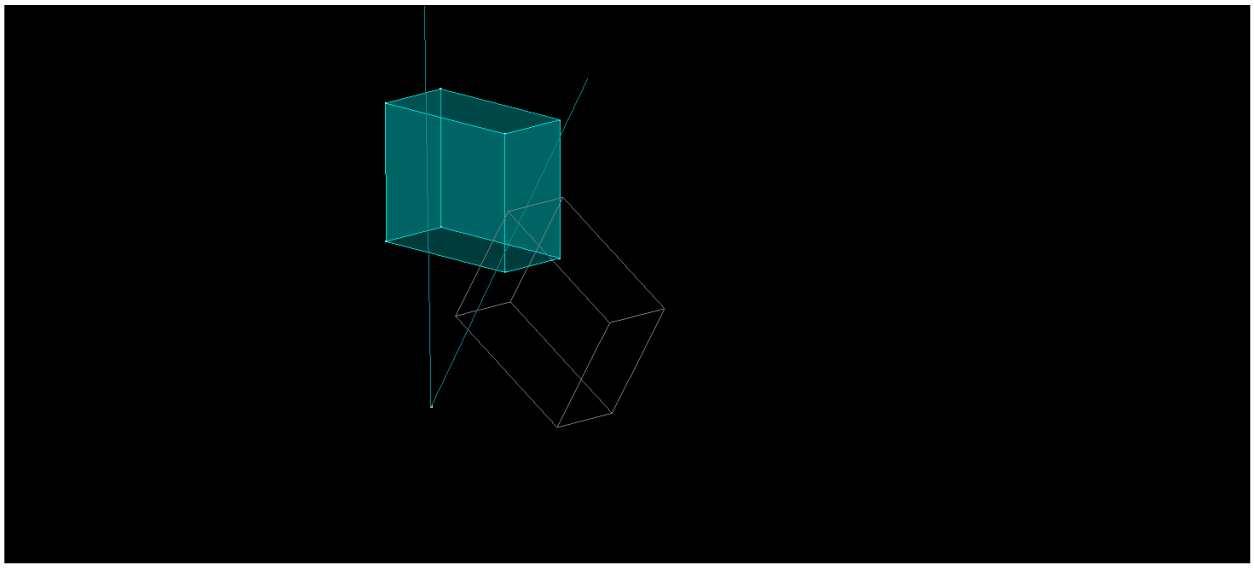
Number of copies - number of copies to be created. If the number is more than 1 the specified number of copies will be created. The distance between the copies will be the same as between the selected elements and the first copy.

Make copy on:

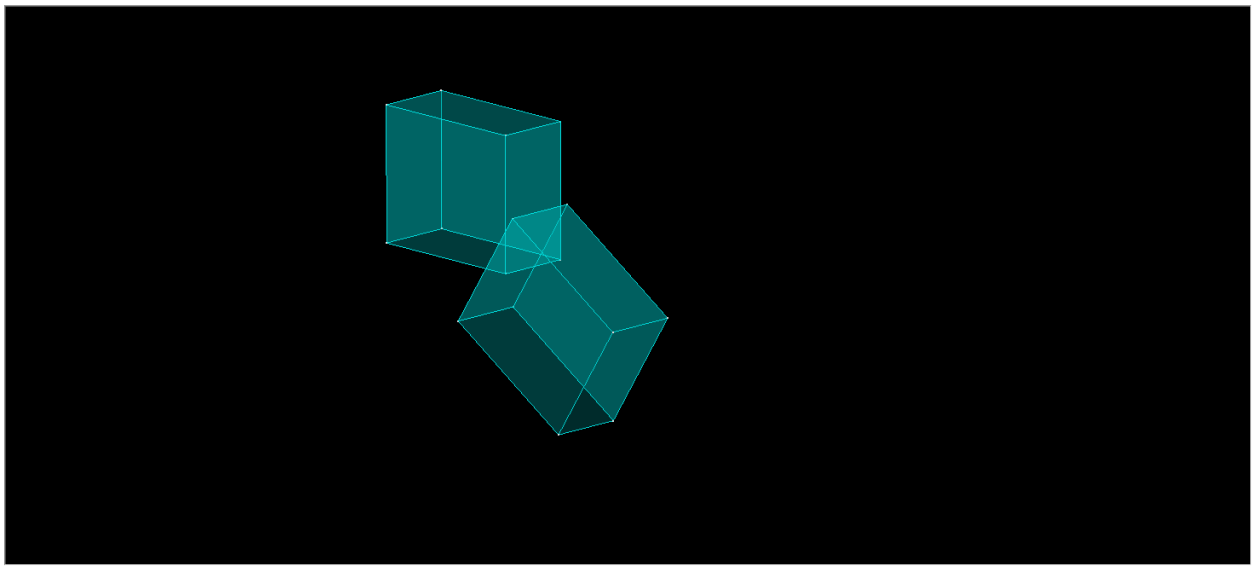
Old support elements - creates a copy with the use of the source reference elements.

New support elements - creates copies without the use of the source reference elements.

Specify the center of rotation and vector of the first angle and second angle.



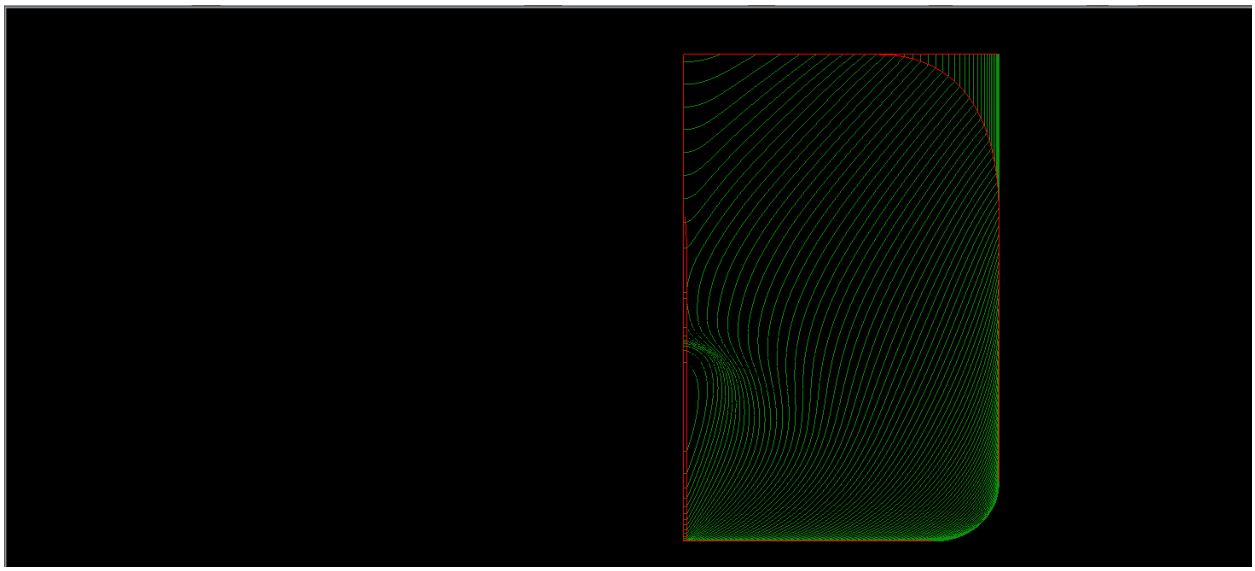
Copied elements will appear in the model.



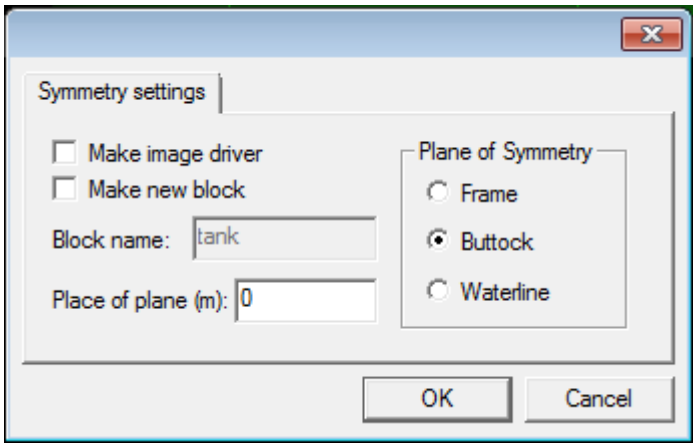
**Block ▸ Symmetry**

This command makes mirror symmetry of the selected elements.

Select the elements to be mirrored. Press Enter after finishing selection.



Select symmetry options in the dialogue box.




Make image driver - crates an image driver.

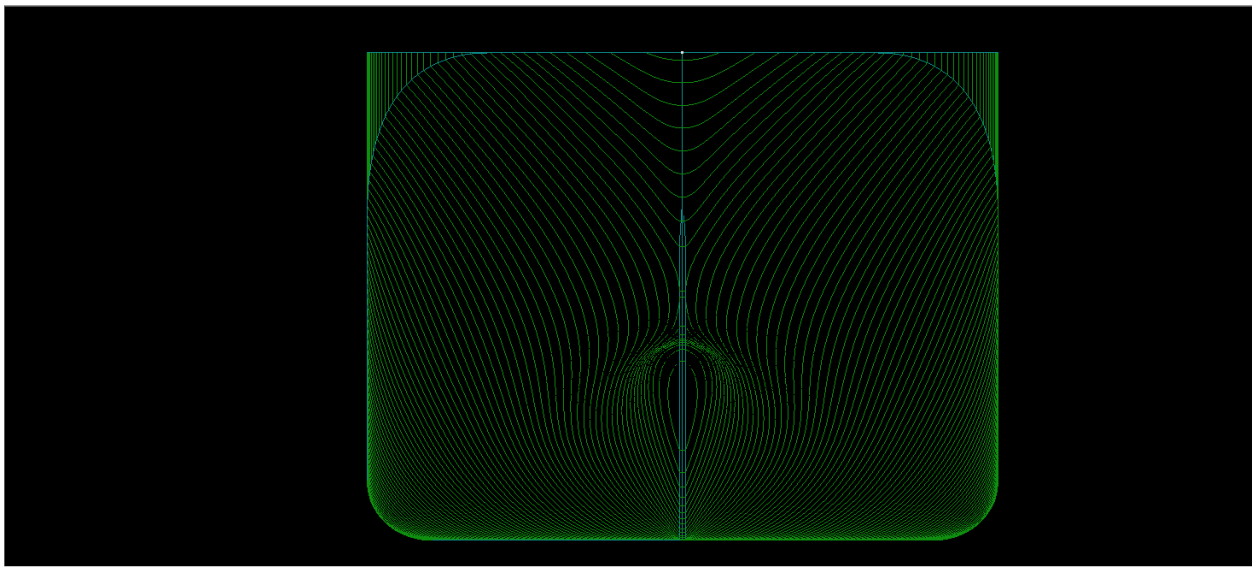
Make new block - creates a block for the copied elements.

Block name - name of the new block.

Place of plane - distance of the symmetry plane from zero to the relevant coordinate.

Plane of Symmetry **-** creation of symmetry relative to planes Frame**,** Buttock**,** and Waterline.

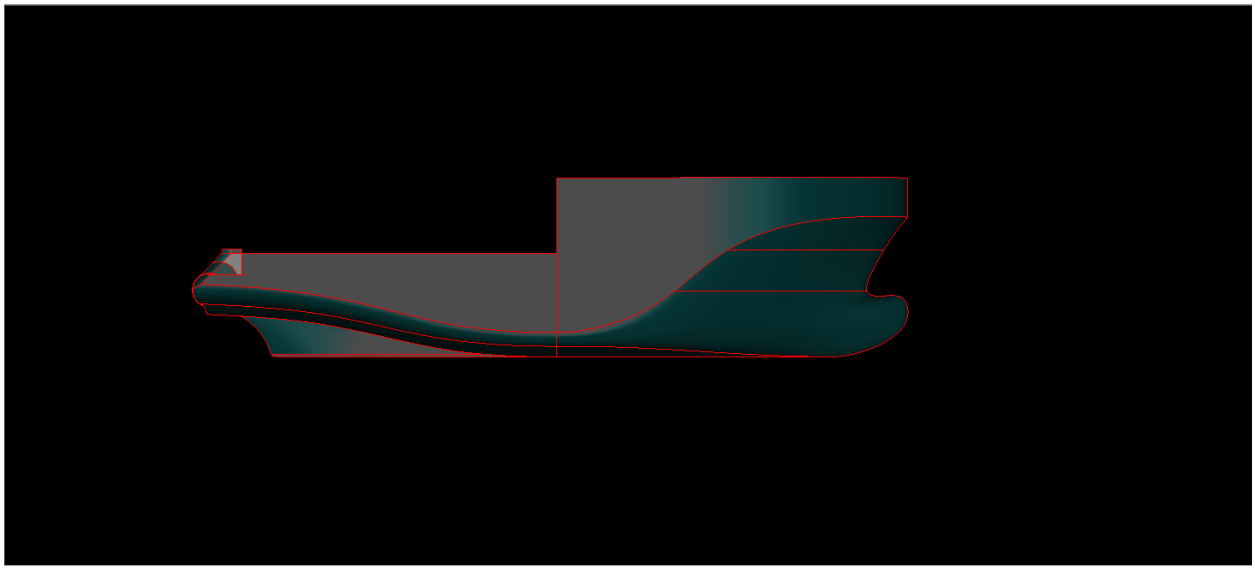
Copied elements will appear in the model.



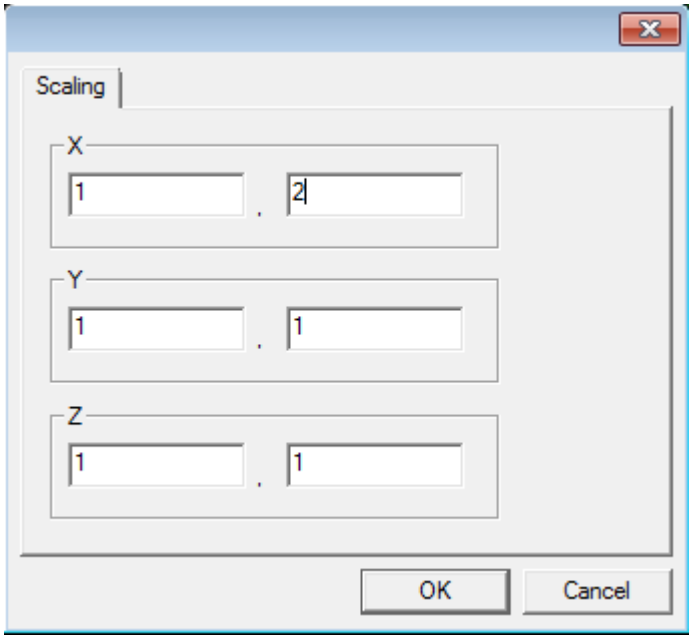
**Block ▸ Rescale**

This command makes scaling of the selected elements.

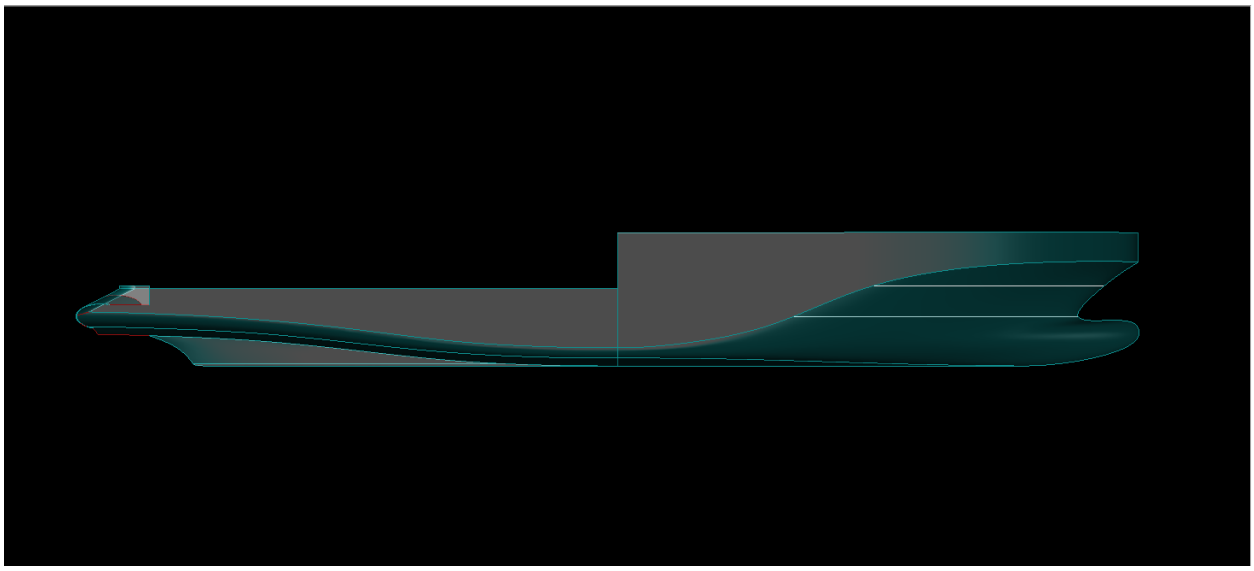
Select the elements to be scaled. Press Enter after finishing selection.



Select basic point of the transformation. Select transformation options in the dialogue box.



Scaled elements will appear in the model.



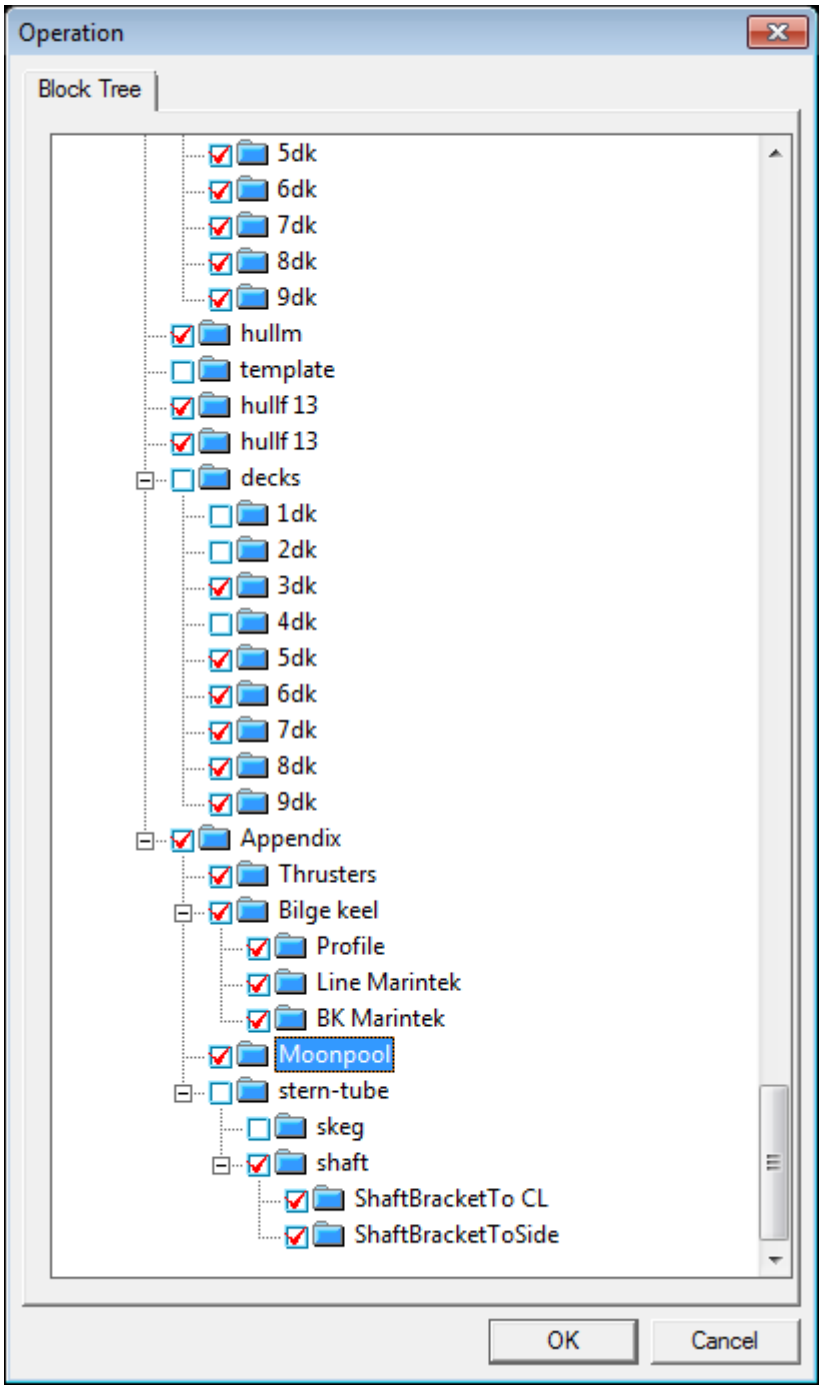
**Block ▸ Set  To Current**

This command change block's attribute of the selected elements. All selected elements will be transferred to the current block.

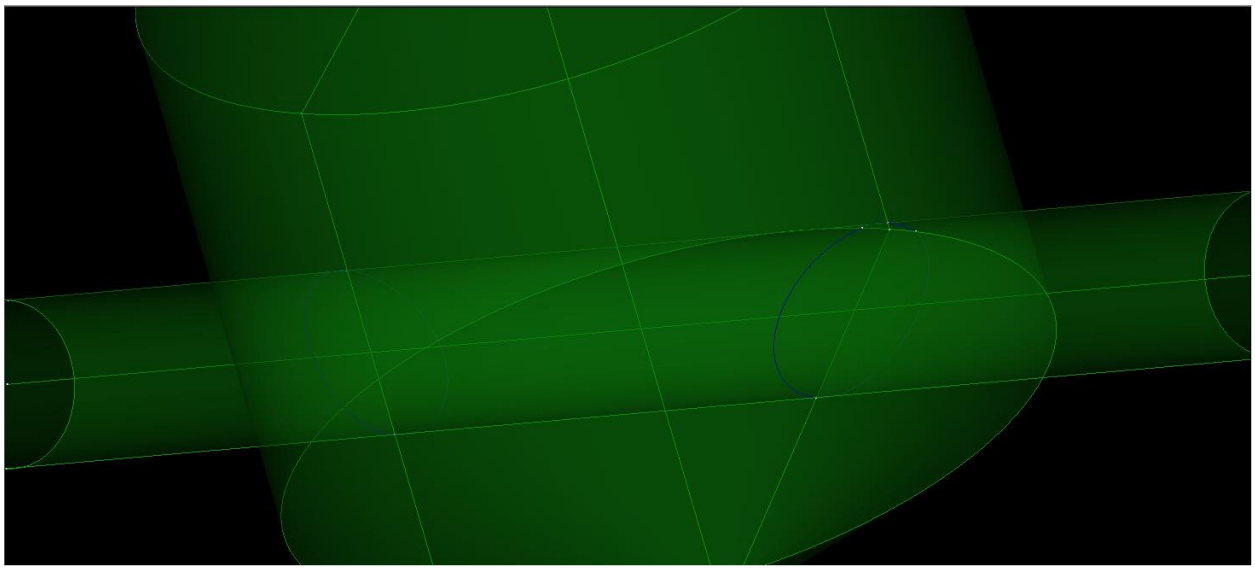
**Block ▸ Intersection**

This command creates intersection lines of all the surfaces included in the intersected blocks.

Select first and them second blocks from block tree:



Intersection lines will appear in the model.



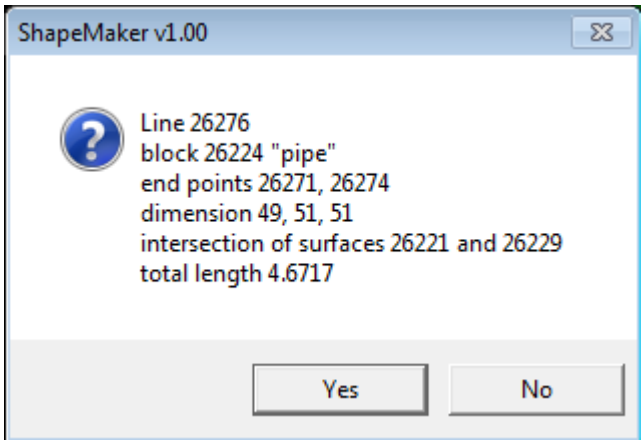
## Modify ▸ Color

Select an element to change its color to the current color of the system.

## Modify ▸ Properties

This command is used to obtain information on a project element.

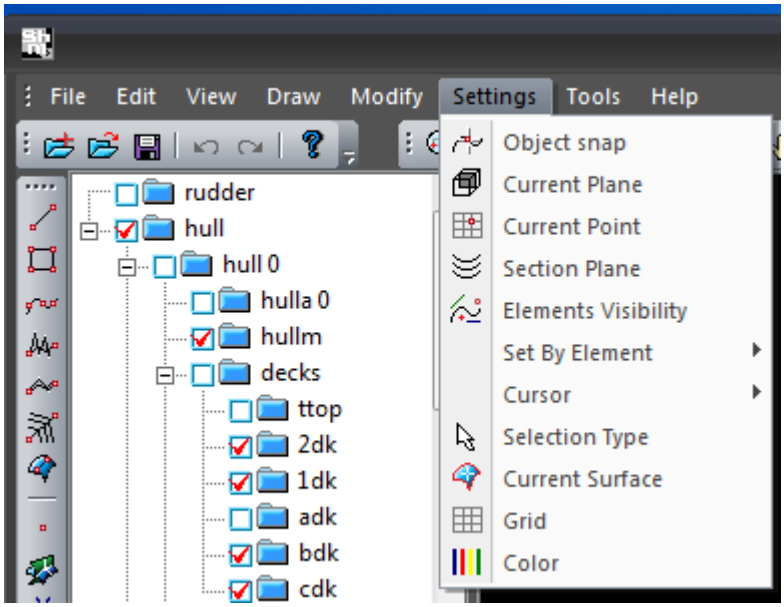
Select element. View information about the element in the dialogue box.




To cancel viewing click No. To view backward references click Yes, you will see a list of elements in the dialogue box.

# Settings

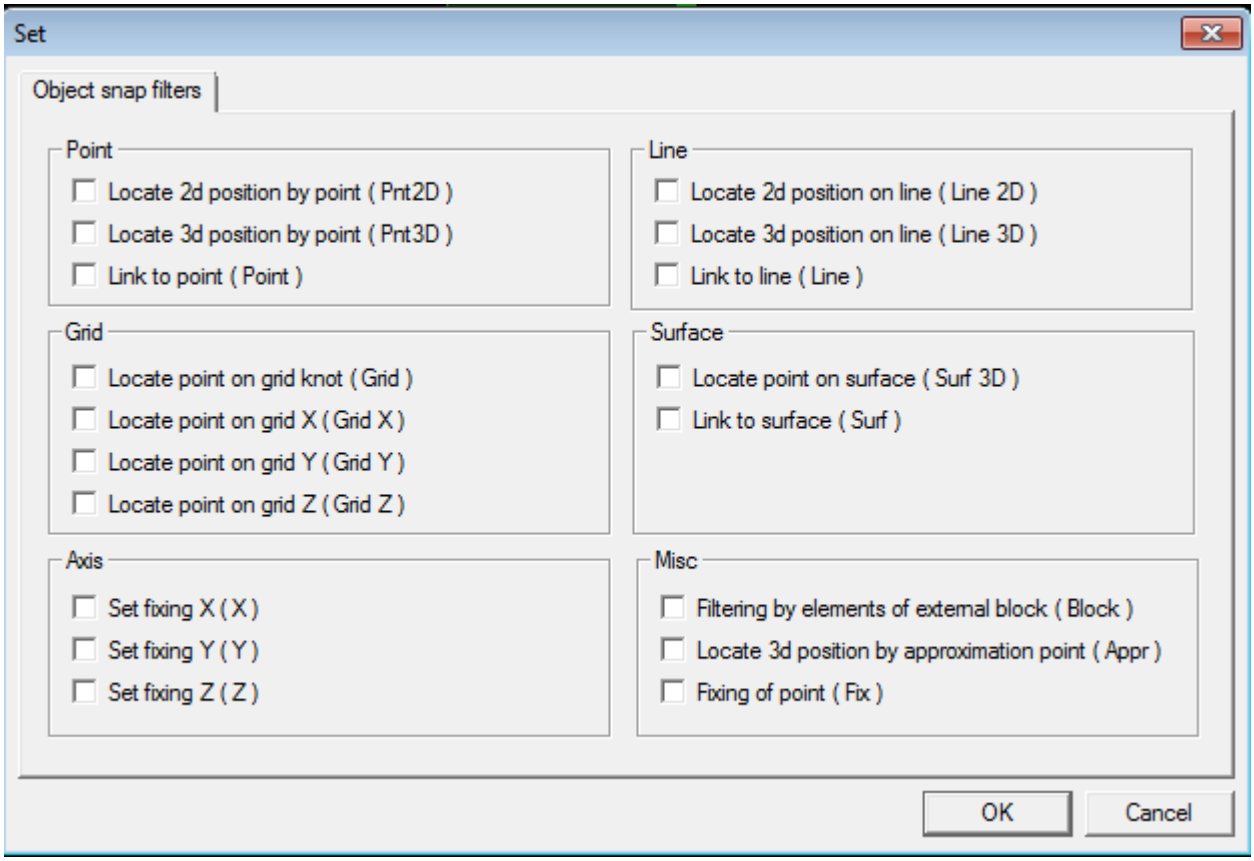
Settings Menu contains commands that allow modify current settings.




## Settings ▸ Object Snap

Command is used to control enabling and disabling the object snaps.

Enable or disable snaps in the dialogue box.



Point:

Pnt2D – geometrical snap to the selected point by two coordinates of the point in current working plane.

Pnt3D – geometrical snap to the selected point in three coordinates.

Point – topological link to the selected point.

Line:

Line2D – geometrical snap to the line in two coordinates in current working plane.

Line3D – geometrical snap to the line in three coordinates.

Line – topological link to the line.

Grid:

Grid – geometric snap to the grid knot.

GridX – geometric snap to the grid in X

GridY– geometric snap to the grid in Y

GridZ– geometric snap to the grid in Z

Surface:

Surf3D – geometric snap to the surface in three coordinates

Surf – topological link to the surface;

Axis:

Snap with assignment of X, Y, Z coordinates.

X – snap in X coordinate;

Y – snap in Y coordinate;

Z – snap in Z coordinate;

Misc:

A snap to more than one object can be simultaneously performed. For example: a snap to the line and to the grid line. For this purpose establish the required combination of the snap modes.

Appr – snap to the approximation point;

It allows performing a geometric snap to approximation points.

Block – snap to the block element;

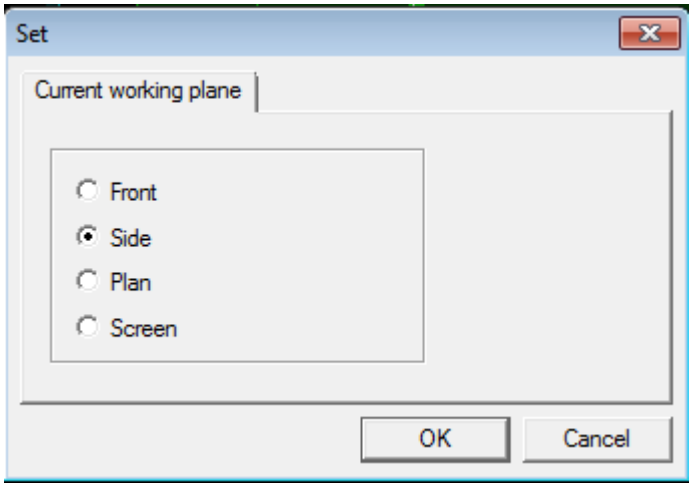
Fix – snap with fixation;

When performing the above said object snap methods the point may assume only few fixation types. Snap with fixation allows assigning any other type of fixation to the point (the same result can be obtained with the help of command **Modify ▸ Edit ▸ Fixing**).

## Settings ▸ Current Plane

This command allows setting the working plane parallel with the main planes in isometrics, i.e. one of the coordinates **X**, **Y** or **Z** is not changed.

Select the work plane in the given dialogue box.




## Settings ▸ Current Point

This command allows changing position of the current work point so as to change position of the work plane.

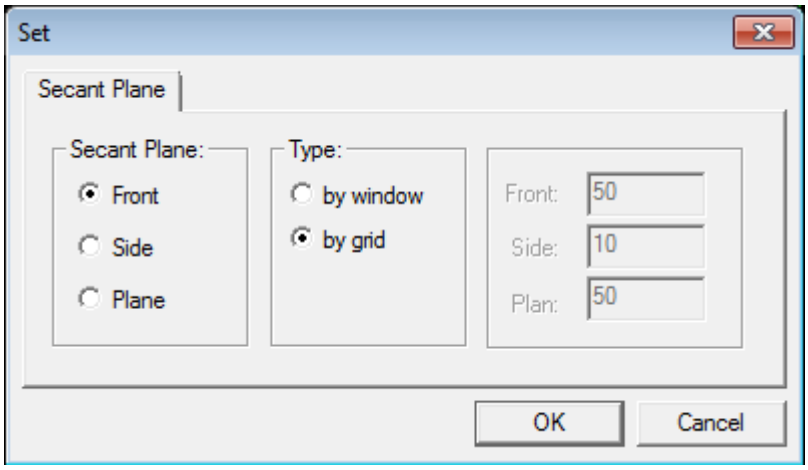
The work point will automatically change its position when carrying out operations for construction of elements. In this case the last input point is saved as the work point, for example, the end of the line.

Input the work point by request.

## Settings ▸ Section Plane

This command is used for the plane and the number of surface sections in planes X, Y, Z to be displayed on the screen.

When the dialogue box appears select the parameters for display of the sections.



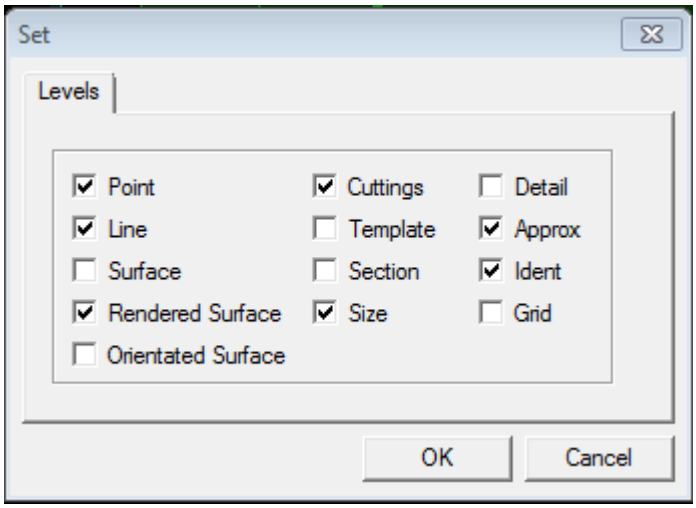
Secant Plane sets a plane of sections for isometric view.

Type sets a number of sections in the grid or by the number that you define in the text fields Front, Side and Plan.

## Settings ▸ Elements Visibility

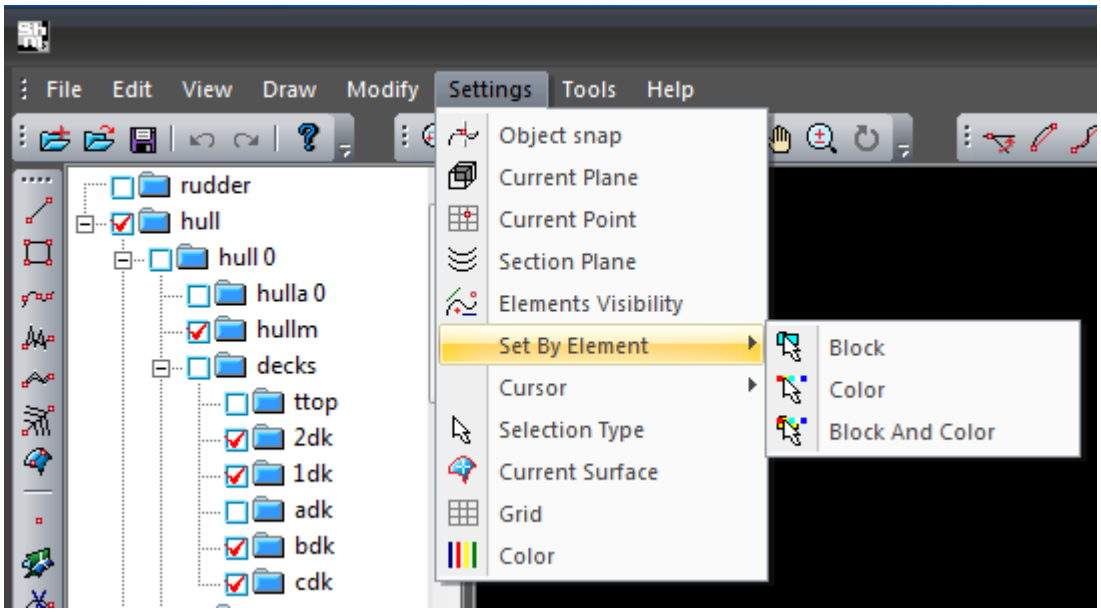
This command is used to control visibility of the elements at the various.

When the dialogue box appears select the elements to be displayed on the screen.




## Settings ▸ Set By Element

Set By Element Menu contains the commands which allow setting current block and color same as selected element.




### Set By Element ▸ Block

This command allows setting a current block by any selected element of the model. A new current block will be the same as block of the selected element.

### Set By Element ▸ Color

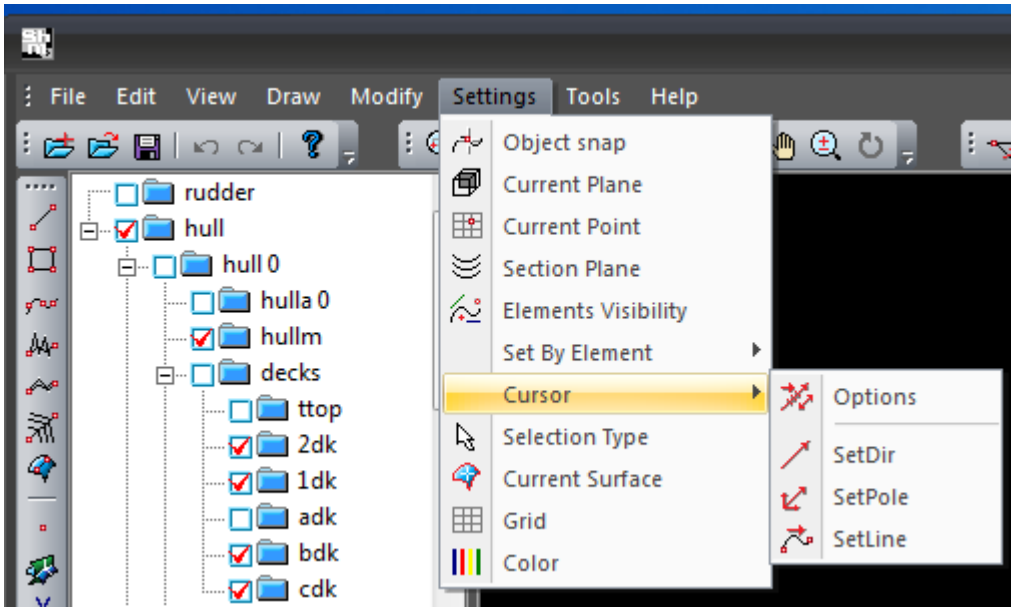
This command allows setting a current color by any selected element of the model. A new current color will be the same as that of the selected element.

### Set By Element ▸ Block And Color

This command allows setting a current block and color by any selected element of the model. A new current block and color will be the same as those of the selected element.

## Settings ▸ Cursor

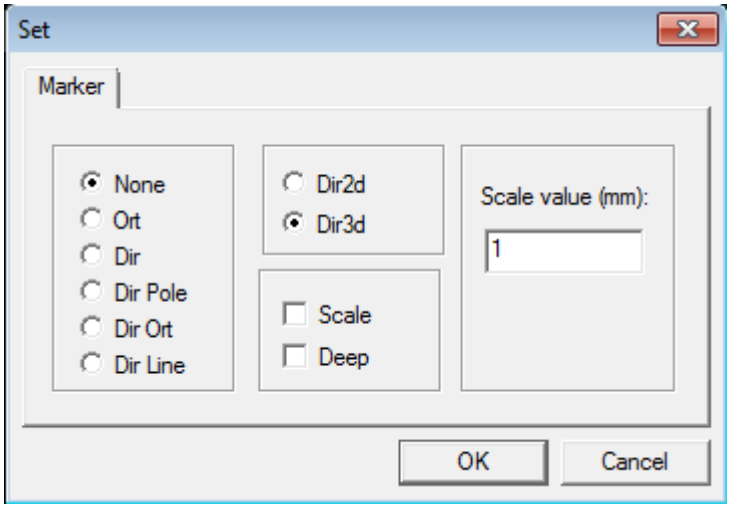
Cursor Menu contains the commands which allow defining marker moving rules.




### Cursor ▸ Options

By this command user can set limitations for cursor movement.

In the dialogue box select required limitation for cursor movement.




Ort - Orthogonal direction on current projection plane.

Dir - Direction predefined by command SetDir.

Dir Pole - Direction predefined by command SetDir and pole predefined by command SetPole.

Dir Ort - Direction predefined by command SetDir.

Dir Line - Direction predefined by command Set Line. Cursor will follow line .

Dir2d - Direction predefined by command SetDir  only in a working plane.

Dir3d - Direction predefined by command SetDir in 3D space.

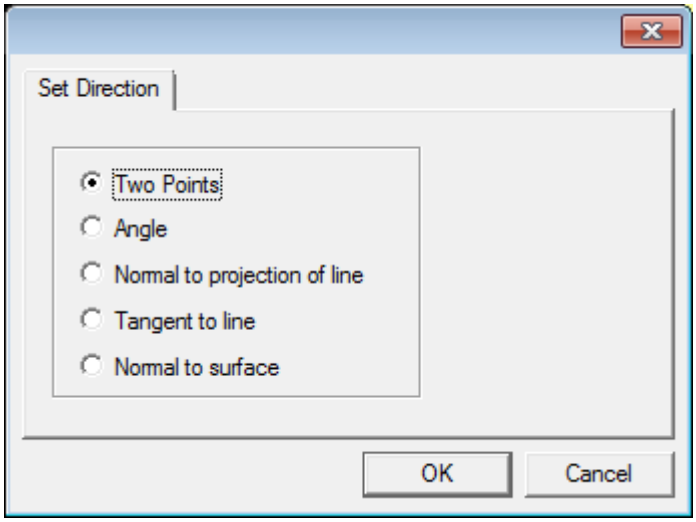
Scale - Enable scalable cursor movement.

Deep - Enable cursor movement in perpendicular to working plane direction.

### Cursor ▸ Set Dir

This command allows specifying a straight line, which defines direction of the point offsetting in the modes Dir, DirPole and DirOrt.

In the dialogue box select the point offsetting direction method.




Two Points – a straight line passing through two points.

Angle - a straight line passing through a point at a specified angle of slope (in the current work plane).

Normal to projection of line – a normal to projection of the frame line on the work plane.

Tangent to line – a tangent to projection of the frame line on the work plane.

Normal to surface – a normal to surface in the specified point.

### Cursor ▸ SetPole

This command is used for input of a point - pole for movement of the point in mode DirPole.

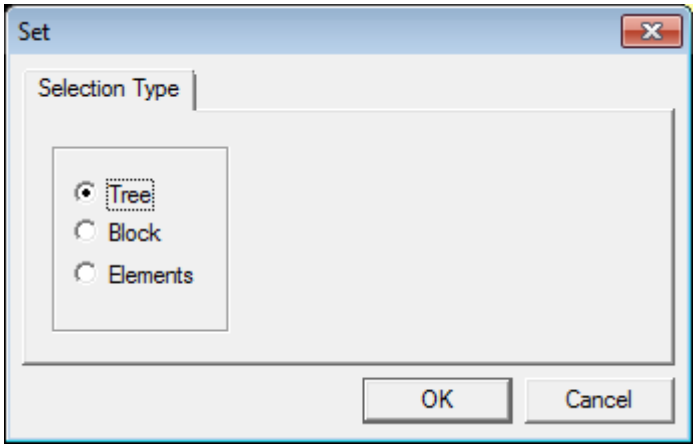
### Cursor ▸ SetLine

This command allows selecting the point movement line in the mode **DirLine**.

## Settings ▸ Selection Type

This command is used to operate with selected elements at the same time (for example: to delete a group of lines without successively specifying each line).

In the dialogue box choose the selection type:




Tree – select block from blocks tree.

Block – select block by any element of the block.

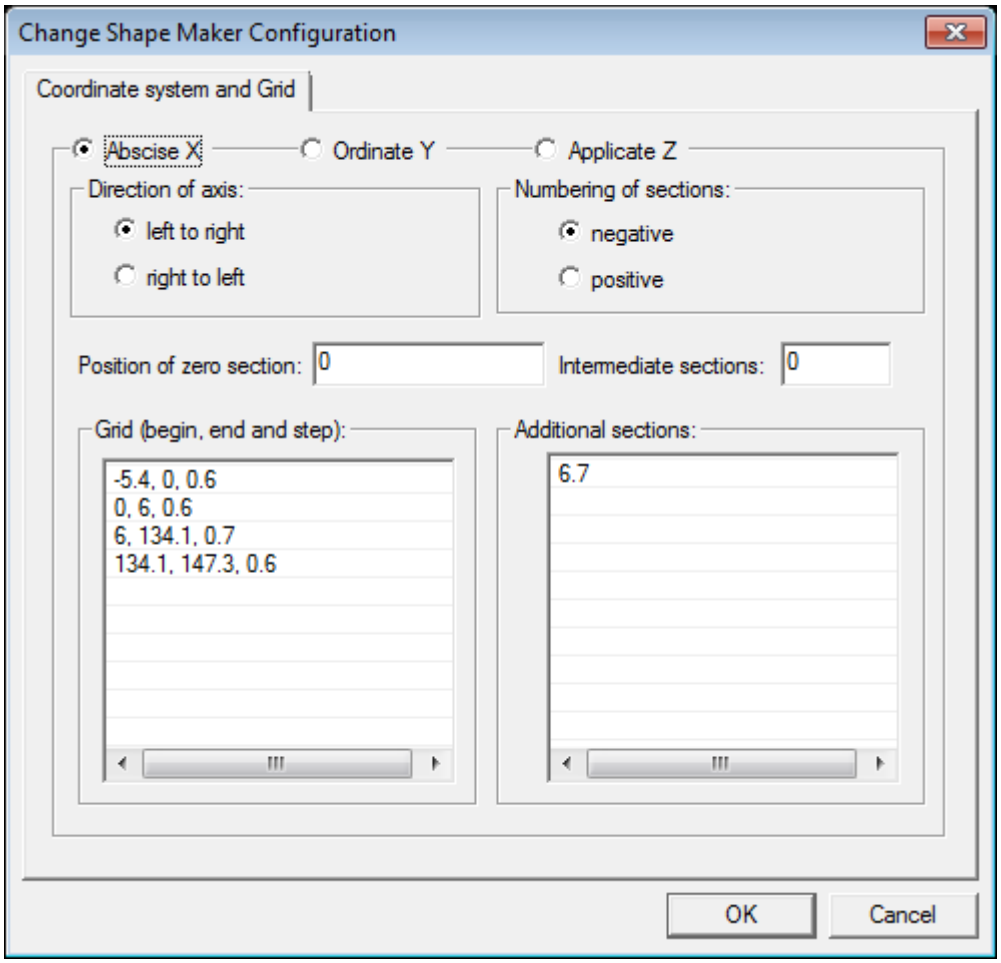
Element - one by one selection of separate elements.

## Settings ▸ Current surface

This command is used to select a default work surface for creation of a lines on surface.

## Settings ▸ Grid

This command is used to specify a User Coordinate System (UCS) and grid of the project Lines Drawing (LD).



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

The grid axis is selected with the help of the keys Abscise (Abscises) X, Abscise Y, Abscise Z**.**

Direction of the grid in axis X is specified from the left to the right or from the right to the left with the help of the following options Direction of Axis: left to right or right to left.

Forward and back direction of the grid in axis Y is specified with the help of the following options Direction of Axis: to you (back) or from you (forward).

Bottom-up or top-down direction of the grid in axis Z is specified with the help of options Direction of Axis: down to up (bottom-up) or up to down (top-down).

Coincident and opposite direction of the numeration of sections with the help of the following options Numbering of Sections**:** negative or positive.

Position of zero section **-** makes possible to set numeration of sections so that the section with number 0 will not coincide with coordinate 0 corresponding to the coordinate system axis.

Intermediate sections – number of sections between grids sections.

Grid (begin, end step) - makes possible to set the beginning, the end and step of the area.

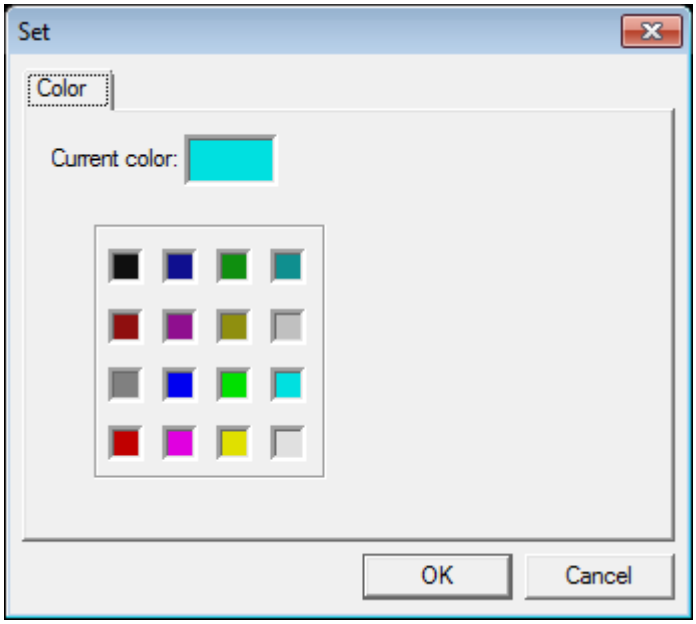
Additional sections - allows additional sections with defined coordinates.

Note:

Interval between begin and end must contain only whole number of steps. Otherwise the error message will be shown.

## Settings ▸ Color

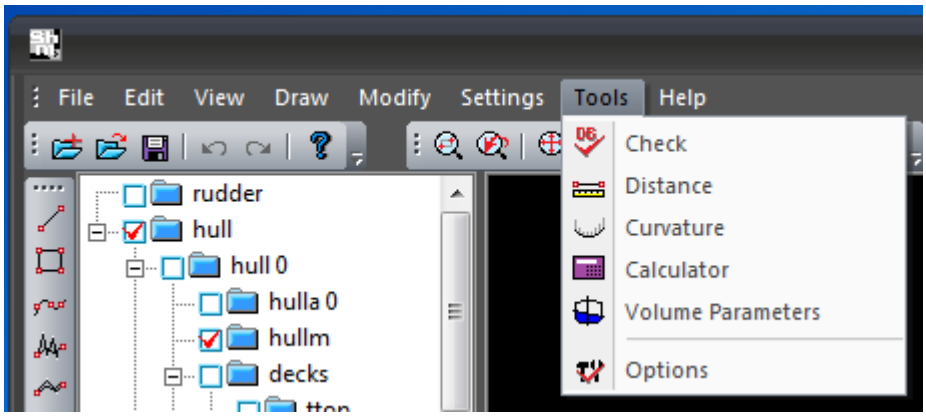
This command is used for setting a current color.




Click on the desired color. The selected color will appear in the window Current color. To confirm the selection click OK. All newly created elements will have the selected color.

# Tools

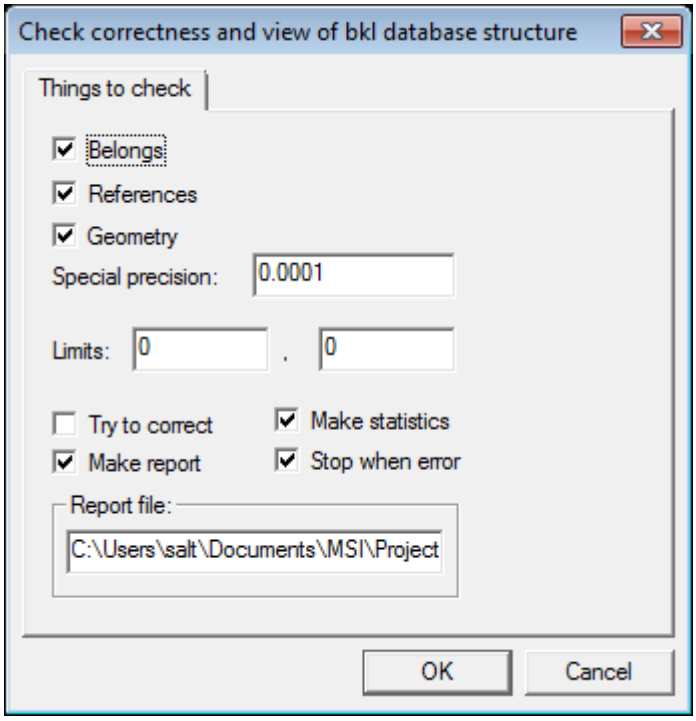
Tools Menu contains commands that allow to make database check, measure distances and angles, use  some auxiliary calculations.




## Tools ▸ Check

This command allows testing the internal database structure, checking dependencies between the elements and properly correcting errors in most of the cases.

Select database checking options in the dialogue box.




Belongs - a feature, indicating a necessity of checking the correctness of “master” for each element. For lines, points and surfaces: affiliation with the block will be checked, and for approximation points – lines or surfaces and etc.

References - indicates a necessity of checking the correctness of references between the elements.

Geometry - the geometry check is checking whether the ends of the lines match their end points and whether the hanging points exactly fit the reference line and etc.

Special precision - database checking precision.

Limits of checking 0, 0 – the range of the checked names. You can narrow the range of the checked names. You can also set the initial and final database element to be checked. The default setting is "0, 0", which corresponds to checking of the whole database.

Try to correct - this option corrects the errors revealed. Corrections may include changing references or the type of an incorrect element, deleting of the latter or rebuilding the geometry.

Make statistics - when this option is enabled the system will make statistics of the data base during the check (total number of elements and of each type, total number of links, average distance between the depending elements, name space fill and etc.). On completion of checking the report file can be read.

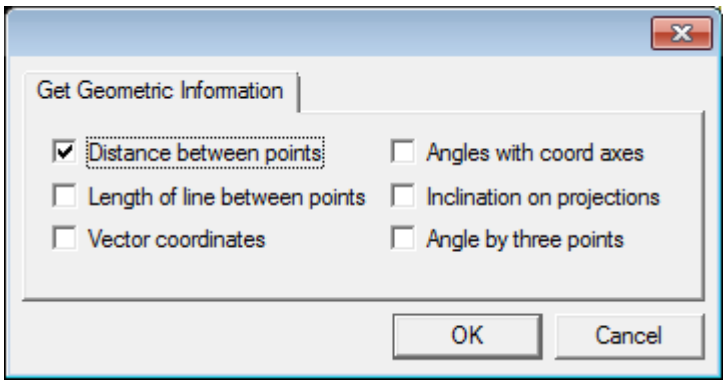
Stop when error - stop when an error occurs. With the help of this option the user can get the information when an error is revealed and take a decision on its correction and whether the check will be continued or terminated.

Make report - this option indicates to the system the necessity of creating a report on the operation. In case of any errors revealed the system will make it possible to view the report file on completion of the check.

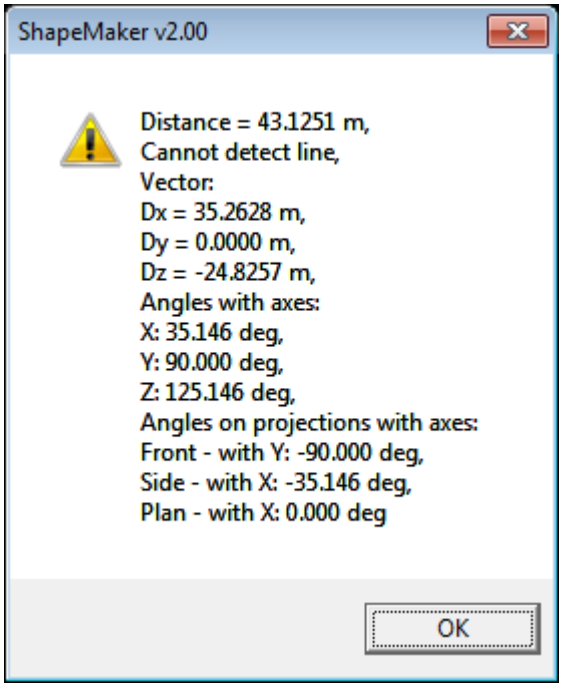
Report file - checkbkl.lst – name of the report file.

## Tools ▸ Distance

This command measures distance between two points, inclination angles of lines, lengths of the liners and other geometrical characteristics.

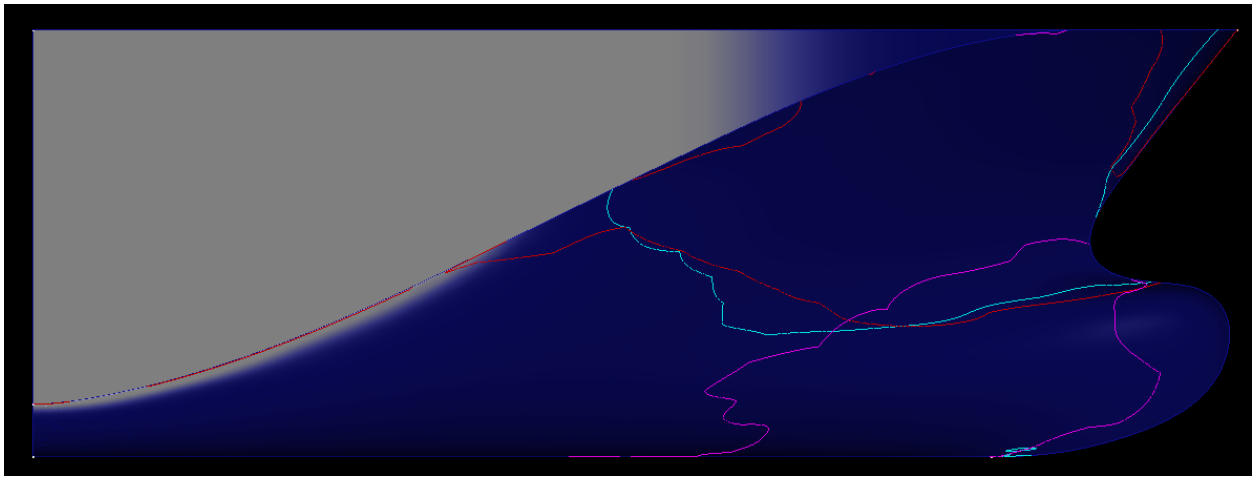



Distance measurement result:



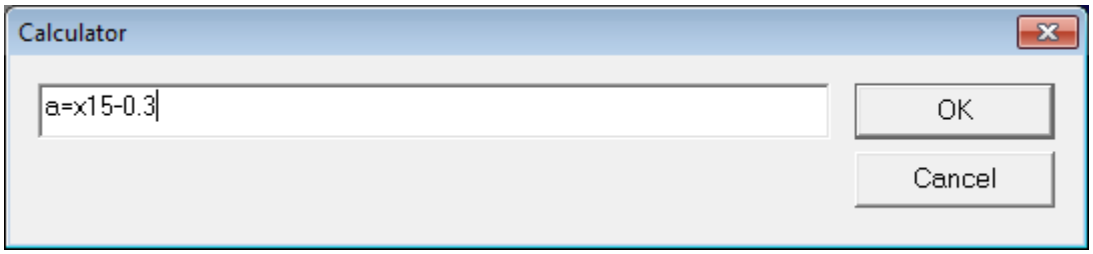

## Tools ▸ Curvature

This command visualizes curvature and inflection lines for selected surfaces.



## Tools ▸ Calculator

The calculator helps to make simple calculations with variables and frame numbers. Variables can be also used in command string for definition of the points coordinates.




The expressions can also be used in the prompt when entering the numerical expressions (for example, when entering a point coordinates). Calculator has collection of the mathematical functions and collection of variables, signed by the Latin letters from a to z. The calculator save variables contents during working session.

In the calculator are available the following operations (in the follow of priority):

, (comma) - means, that previously calculating value from the left and after that – the result from the right from it (for example the expression “a = sin(x), a + a” will give value 2*sin(x)). Except it comma used for dividing setting numbers and arguments of the functions;

= equalization;

+ , - addition and subtraction;

* , / , % multiplication, division and the rest from division on the whole number, correspondingly;

^ raising to the power.

Mathematical functions:

abs ( x ) - |x|,

acos ( x ) - arccos x,

asin ( x ) - arcsin x,

atan ( x ) - arctg x,

cos ( x ) - cos x,

ch ( x ) - ch x,

exp ( x ) - ex,

int ( x ) - [x],

ln ( x ) - ln x ,

lg ( x ) - log10 x,

sin ( x ) - sin x,

sh ( x ) - sh x,

sqrt ( x ) - √X,

tg ( x ) - tg x,

th ( x ) - th x,

rad ( x ) - x× p /180,

deg ( x ) - x× 180/p

Predefined variables:

a = 1/137,

c = 2.997925*1010,

e = 2.71828182845904523536,

f = 2.9*1014

g = 6.672*10-23,

h = 1.0546*10-27,

k = 1.38*10-16,

m = 0.911*10-27,

n = 6.0222*1023,

p = 3.14159265358979323846,

q = 4.803*10-10,

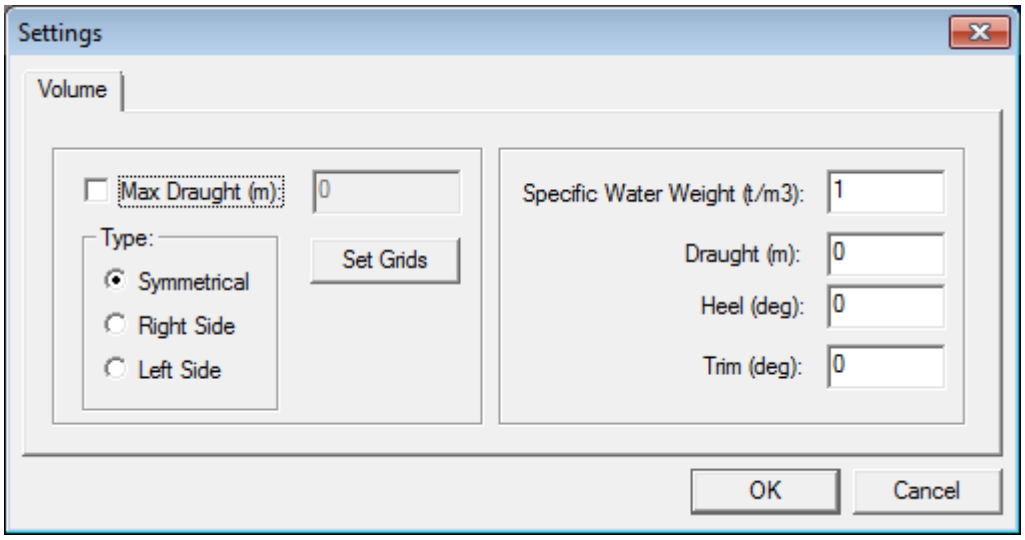
r = 8.314*107,

s = 0.577215664.

Other variables defined by zero.

## Tools ▸ Volume parameters

Simple hydrostatics calculations can be performed by this command.




Max. Draught – maximum loaded draught

Type:

Symmetrical – symmetrical.

Right side - starboard.

Left side - port side.

Set Grids - formation of the own grid ( only frames spacing can be defined) .

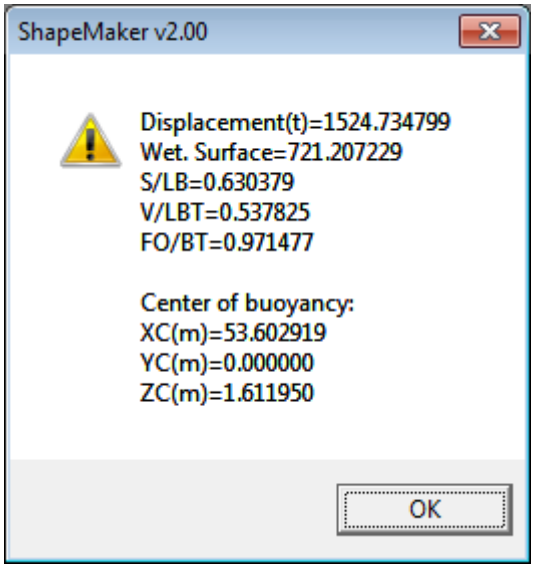
Specific Water Weight (t/m3) – water density

Draught (m) – draught

Heel (deg) – angle of heel

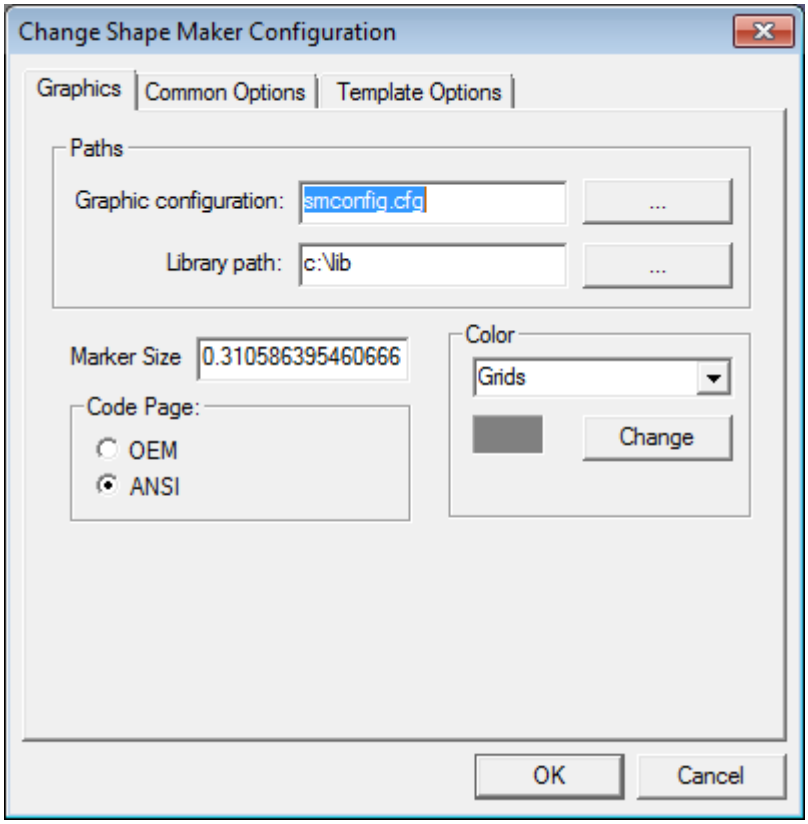
Trim (deg) – angle of trim

Results of the calculations:




## Tools ▸ Options

Simple hydrostatics calculations can be performed by this command.




Graphic configuration - smconfig.cfg – name of the file which contains information on the graphics options (by default - smconfig.cfg).

Library path - c:\ lib – directories where the library files will be searched. You can set several paths divided by a semicolon.

Color:

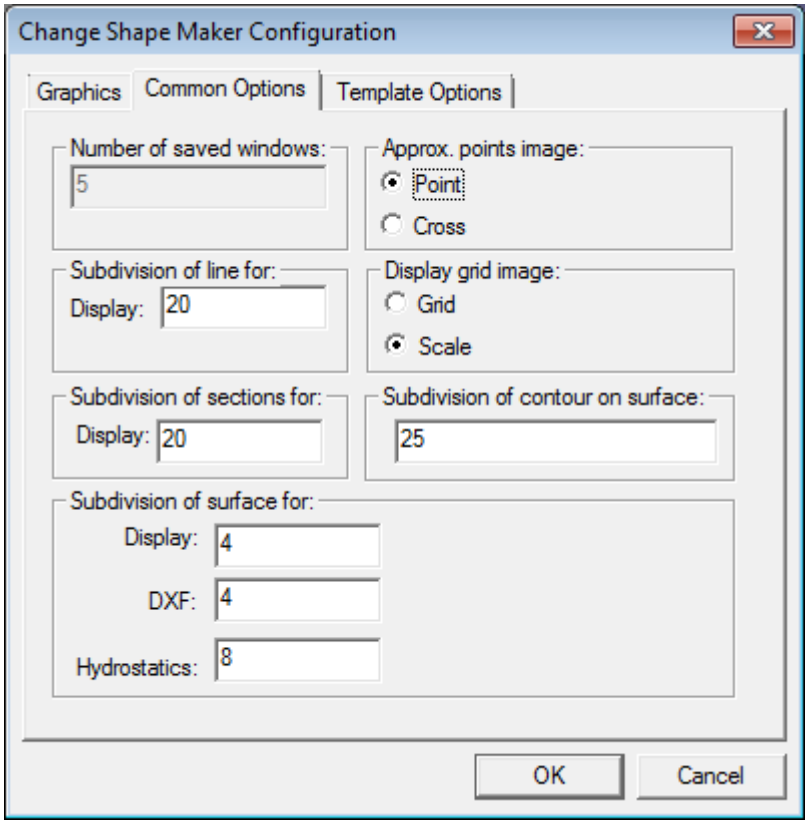
Grid – Grids Color

Window Background - Window Background Color

Code page - selection of coding.

ANSI –ANSI Windows coding page

OEM – OEM coding page




Number of saved windows - 5 – number of windows to be saved (from 2 to 5) for command Window ▸ Prev.

Subdivision of line for Display - line representation quality (number of line points per Bezier segment) to be displayed on the screen.

Subdivision of sections for Display - size of matrix of sections displayed.

Increase size of the matrix for better display of sections.

Subdivision of contours on surface - 25 – representation of cutout borders on the surface (similar to representation of lines).

Subdivision of surface for - size of matrix of surface displayed (number of lines of equal parameter):

Display - 4 – for displaying on screen,

DXF - 4 – for creation of DXF file.

Hydrostatics - 8 – for creation of STC statics file.

The more is the number of points, the better is the quality of representation of lines. Besides, it slows down the display and increases size of the DXF - file;

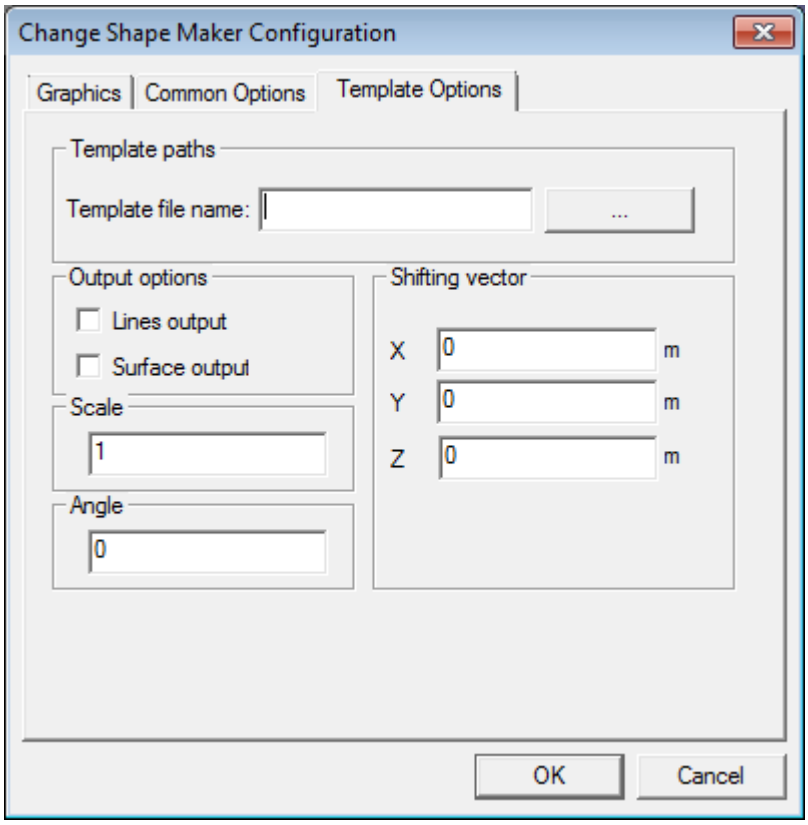
Approximation points image - quality of representation of approximation points.

Point - as points,

Cross - as asterisks.

Display Grid image: - grid display type.Grid - as grid,

Scale - as a contour of lines limiting the volume and marks in such lines.




Template options used for visualization of CFD results and other 3D model saved in files TPL.

Template file name – name of the file for visualization.

Output options:

Lines output – visualization of the lined ( for example stream lines )

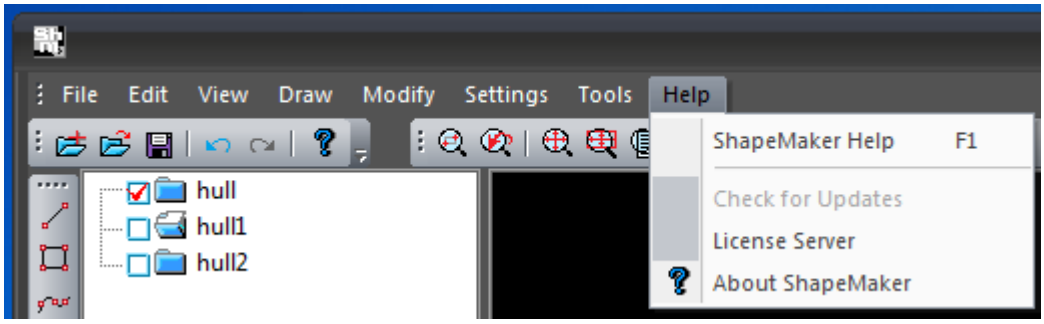
Surface output – visualization of the surfaces like wave pattern.

Scale – scale of the model.

Angle – trim angle of the model.

Shifting vector – defined possible shift of the model 0 position.

# Help




Contain help information, license server connection and release data.

## Help ▸ ShapeMaker help

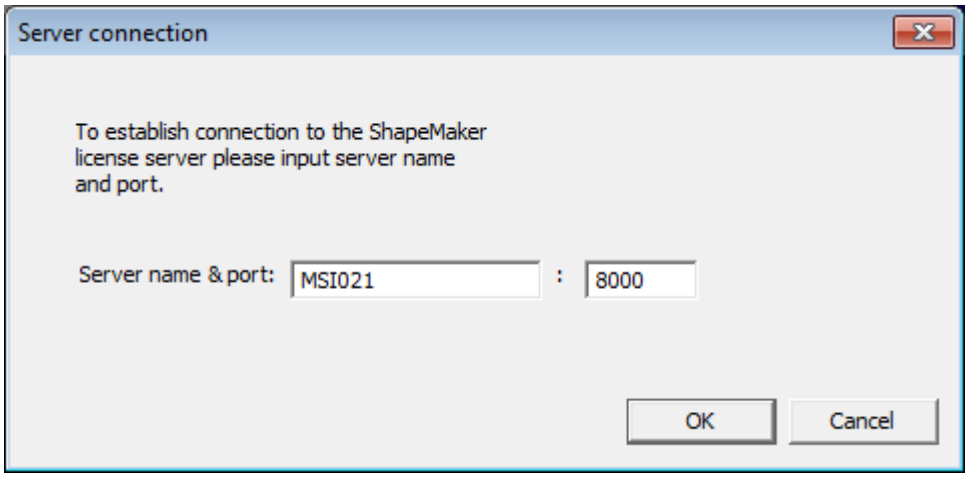
Show this manual.

## Help ▸ Check for update

Provide check for new release on Shape Maker release server. ( For registered users only ).

## Help ▸ License server

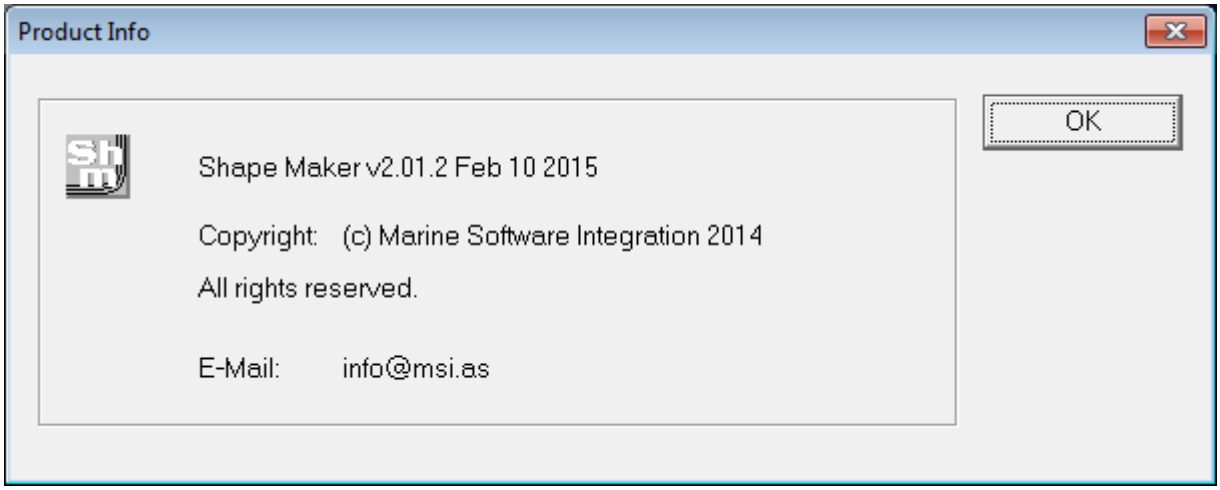
Provide connection to the license server.




Input server name or IP address and port number.

## Help ▸ About ShapeMaker

Show this window.


Product Info
OK
Shape Maker v2.01.2 Feb 10 2015
Copyright: (c) Marine Software Integration 2014
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E-Mail: info@msi.as